

AnTherm Help

AnTherm[®]

Analysis of Thermal behaviour of Building Constructions with Heat and Vapour Bridges

AnTherm[®] is a powerful application used for the "*analysis of thermal behaviour of building components with heat and vapour bridges*". It calculates *temperature distribution* and *heat flows* and/or *vapour diffusion flows* in building structures of arbitrary form and complex material composition - particularly such with thermal bridges. It presents *critical condensing air humidity (dew point)* at all component's surfaces and shows the distribution of *partial vapour pressure* in component's interior. This application is also suitable for analysing *dynamic behaviour* of building components (for *periodic, harmonic, transient* boundary conditions).

For the technically qualified designer, AnTherm is a reliable, indispensable tool in meeting the demands of the current European standards (EN) for evaluating thermal performance thoroughly and precisely.

In particular the program allows the calculation of:

- steady state (stationary) and transient calculation of temperature distribution of components in [two- and three](#),
- fully automatic calculation of [the matrix of thermal coupling coefficients L2D or L3D](#),
- completely automated calculation of [linear thermal transmittance \$\Psi\$ \(psi\)](#)
- transient periodic, harmonic calculation of [the matrix of dynamic \(harmonic\) coupling coefficients](#) (with the HARMONIC-Option),
- fully automatic calculation of [temperature weighting factors](#) (fRsi, g) at coldest surface points of every spaces,
- equally appropriate for very detailed modelling (e.g. facade constructions, window frames, ...) and for very large calculation cases (e.g. components at ground surface, complete spaces, groups of spaces, ...),
- calculation and presentation of critical condensing air humidity (dew point) at all component's surfaces,
- calculation and presentation of partial vapour pressure and, by comparing it to the saturation pressure, simple identification of condensation risk in component's interior (with the [VAPOUR-Option](#)),
- extensive [three dimensional graphical evaluation \(visualization\)](#) also dynamically, under [time dependant, periodic boundary conditions](#).

AnTherm[®] is fully [validated according to EN ISO 10211](#): AnTherm has been qualified as a "Class A precision tool" – as for two- and three-dimensional, steady state precision method. It also [fulfils all requirements raised in EN ISO 10077](#) for the simulation program to qualify it as standard method for calculation of heat flow through frames of windows, doors and shutters.

Daily experience gained so far by using the program AnTherm[®] have impressively proven, that the input, calculation and the [evaluation part](#) of the program satisfy highest demands regarding highest complexity of calculated components.

Very high resolution (e.g. modelling at 1/10 mm ranges or less) as well as very large models (e.g. complete building assemblies or components in contact with ground) can be easily dealt with.

Major strengths of that software, like unlimited number of elements, easy to control and flexible graphical evaluations even with very large computational cases, direct calculation of thermal coupling coefficients and temperature weighting factors, etc. have been implemented very carefully in AnTherm[®].

Essential properties of the program

With respect to input data, *AnTherm*

- facilitates the generation of geometrical models by supporting a graphic input display of building structures, as well as by providing an entirely independent, fully automatic method of fine subdivision (which can also be influenced by the user through manual manipulation of parameters).
- delivers complete input model documentation (geometry of material elements, thermal design values of materials, spaces, heat sources...) upon request.
- allows the precision of numerical solutions to be influenced and controlled by the user (definition of calculation parameters).

Evaluation results yielded by *AnTherm* are conformant to European Standards and include:

- generally applicable results in the form of **g-values** and **conductance matrices**. These conform to the **temperature weighting factors** and "**thermal coupling coefficients**" defined in the European Standards, including the required information on calculation precision.
- specific results, applicable to particular air temperature conditions in spaces thermally coupled by the building components analyzed, in the form of **surface temperature minima and maxima** as well as respective **dewpoints**.
- graphic plots and prints of **isotherms**, **surface temperatures** or **temperatures along an edge** (2- or 3-dim. models), as well as **heat flow diagrams** (not limited to 2-dim. models only, but **also in 3D**).

Furthermore, **fully automatic execution of calculation** with *AnTherm* is given even in the event of poorly conditioned calculation cases. The maximum quantity of balanceable cells is nearly unlimited (many millions), thus making the thorough analysis of large, three-dimensional models feasible. Complex cases are given, for example, by

- components or spaces in **contact with ground**.
- **entire spatial envelopes** or groups of spaces.
- detailed modelling of complicated assemblies (**window frame and installation details**, steel structures, etc.).

The thermal performance of building assemblies with overall dimensions of up to app. 100 m can be simulated with *AnTherm*.

Limitations to the scope of evaluation depend on the [PC hardware](#) implemented.

Table of contents

AnTherm Help

- [Table of contents](#)
- [Copyright](#)
- [License Terms and Conditions of Use](#)
 - [Acknowledgments and Additional Terms of Use](#)
- [Validation according to EN ISO 10211:2007](#)
- [Validation according to EN ISO 10077-2:2003](#)
- [System requirements](#)
- [Short reference](#)
- [Installation of the software](#)
 - [Installation of MS .NET Runtime and further prerequisite components](#)
 - [License file AnTherm.HID](#)
- [Introduction](#)
 - [Two- and Three-Dimensional heat flow Patterns](#)
 - [European Standards on Thermal Heat Bridges](#)
 - [Structure of the program](#)
 - [The input branch](#)
 - [Elements of the building component](#)
 - [The Building Component](#)
 - [Element overlapping](#)
 - [The Space \(Room\) and the Surface](#)
 - [The heat source \(power source\)](#)
 - [Creating the Grid \(Gridding\)](#)
 - [The Calculation \(Solver\)](#)
 - [The Over-Relaxation](#)
 - [Evaluation of results](#)
 - [Visualization - Graphical Evaluation](#)
 - [Multidimensional Vapour Diffusion](#)
 - [Harmonic \(periodic\) dynamic indicators](#)
 - [Transient \(periodic\) time dependant simulation](#)
 - [Computational performance enhancement for multiprocessor systems](#)
 - [Immersive 3D stereo \(binocular\) experience](#)
- [Country specific localisation](#)
- [Starting AnTherm](#)
- [Project types](#)
- [Coordinate system](#)
- [Results and Evaluation](#)
- [Primary Concepts](#)
 - [Environmental Factors](#)
 - [Thermal Transmission](#)
 - [Conductance](#)
 - [Linear and Point Transmittance](#)
 - [Surface Temperatures](#)
 - [Weighting Factors](#)
 - [Graphic Representation](#)
 - [Graphic related to Psi-Evaluation](#)
 - [Method of Analysis](#)
 - [Calculation Model](#)
 - [Numerical Solution](#)
- [Main application window](#)
 - [Main menu](#)

- [Editing windows \(Input windows\)](#)
 - [Description - window](#)
 - [Element selection \(browser\) - window](#)
 - [Elements 2D - window](#)
 - [Element Editor](#)
 - [Context menu of element editing](#)
 - [Layer selection window](#)
 - [Layer Editor window](#)
 - [Materials & Surfaces window](#)
 - [Materials window](#)
 - [Material Editor window](#)
 - [Materials Database window](#)
 - [Baustoffkatalog ON V 31](#)
 - [Material Database baubook](#)
 - [Material Database BPHDB.COM](#)
 - [Material Database ArchiPHYSIK \(APH\)](#)
 - [Material Database ECOTECH](#)
 - [ECOTECH Material-Catalogue window](#)
 - [Surfaces window](#)
 - [Surface Editor window](#)
 - [Transforms \(Rotate, Mirror, Translate, Stretch, ...\) window](#)
 - [Translate window](#)
 - [Stretch window](#)
 - [Rotate / Mirror window](#)
 - [Rotate window](#)
 - [Mirror window](#)
 - [Slope/Roundness](#)
 - [Elements 3D window](#)
 - [Elements 23 window](#)
 - [Status Report window](#)
 - [Boundary Conditions window](#)
 - [Periodic / Harmonic Data Editor](#)
 - [Probe Points / Extreme \(min/max\) Locations window](#)
 - [Interface AnTherm/SolRad \(HARMONIC/TRANSIENT\)](#)
 - [Psi-Value Calculator \(Tool\)](#)
 - [Psi-Value Calculator for Three Rooms](#)
 - [Psi-Value \(window integration\)" \(tool\)](#)
 - [Usb Calculator](#)
 - [U-Value Calculator \(Tool\)](#)
 - [Chi-Value Calculator \(Tool\)](#)
 - [Inhomogenous Layer Calculator \(Tool\)](#)
 - [Air Cavity Calculator \(Tool\)](#)
 - [Condensing Humidity \(dew point\) Calculator \(Tool\)](#)
 - [Expression Evaluator \(Tool\)](#)
 - [Interface AnTherm/SolRad \(stationary\)](#)
- [Evaluation & Results windows](#)
 - [Timelines Window](#)
 - [Report builder](#)
 - [Solver window](#)
 - [Psi-Value Determination \(Calculate \$\Psi\$ -Value\)](#)
 - [Boundary Conditions window](#)
 - [Periodic / Harmonic Data Editor](#)
 - [Animate Time Dialog](#)
 - [Surface Temperatures Window](#)
 - [Results 3D window](#)

- [General \(control panel\)](#)
- [Results3D Parameter window](#)
- [Model \(control panel\)](#)
- [Surface \(control panel\)](#)
- [Slice X, Y, Z \(control panels\)](#)
- [Isolinies / Isotherms \(control panel\)](#)
- [Labels \(control panel\)](#)
- [Isosurface \(control panel\)](#)
- [Streamlines \(control panel\)](#)
- [Vectors \(HedgeHog, Arrows\) \(control panel\)](#)
- [Colorbar \(control panel\)](#)
 - [Colour scale / Colour tables](#)
- [Probe \(control panel\)](#)
- [Profile \(control panel\)](#)
- [Axes \(control panel\)](#)
- [Outline \(control panel\)](#)
- [3-D Navigation \(control panel\)](#)
- [Isometries window](#)
- [Active \(setting\)](#)
- [Opaque and Opacity \(setting\)](#)
- [Colorize \(setting\)](#)
- [Fixed Range \(setting\)](#)
- [Probe Points / Extreme \(min/max\) Locations window](#)
- [Results3D Parameter window](#)
- [Isometries window](#)
- [Dialog windows](#)
 - [Snap-To-Grid parameter](#)
 - [Assign To Group Dialog](#)
 - [Fine-Grid parameter](#)
 - [Solver parameter](#)
 - [Revolve to 3D Params - dialog](#)
 - [About dialog window](#)
 - [Simulation folder cleaner](#)
 - [Please Wait window](#)
 - [Tip Of The Day](#)
 - [Settings - various \(dialog\)](#)
 - [Leitwert Lambda Equivalent Calculator \(Lambda-EQ Tool\)](#)
- [Further control elements](#)
 - [Toolbar of a Report window](#)
 - [3D Navigation within 3D windows](#)
 - [Trackball and Joystick Rotation](#)
 - [Solid or Wireframe model \(setting\)](#)
 - [Background colour \(setting\)](#)
 - [More Light \(setting\)](#)
 - [Copy image to clipboard \(command\)](#)
- [Working with files](#)
 - [User specific Logo file](#)
 - [Standard project template file projekt.xml](#)
 - [Initial template of window positions AnTherm.exe.LayoutPersistence.xml](#)
 - [Colour setting file LambdaToColor.ColorList](#)
 - [Import of WAEBRU component file](#)
 - [Import of Heat2 input file](#)
 - [Import of Heat3 text input file](#)
 - [Import of Kobru86 file](#)
 - [Import of DXF file](#)

- [Requirements to import DXF file](#)
 - [Import of 3D DXF / IFC File](#)
 - [Import/Export of Excel \(XLS\) File](#)
 - [Import of Image file \(Image Underlay\)](#)
 - [Export to CSV file](#)
 - [Export image to file](#)
 - [Export 3D scene to file](#)
- [Application Settings \(dialog\)](#)
 - [Settings file AnTherm.exe.Settings.xml](#)
- [Theoretical background](#)
 - [On calculation of \$\Psi\$ -values \(Psi\) for a 3 space case: Special consideration: Unheated Spaces](#)
 - [Heat Bridges by Heindl, Kreč, Panzhauser, Sigmund](#)
 - [On condensing humidity \(dew point\)](#)
 - [Calculating equivalent thermal transfer coefficient of air cavities](#)
 - [On calculation of \$\Psi\$ -values for building constructions in connection with ground](#)
 - [Temperature factor fR_{si}](#)
 - [Multidimensional vapour diffusion processes](#)
 - [The air temperature in the unheated basement, a case study](#)
 - [Heat losses from large buildings through the ground](#)
 - [Methods of thermal heat bridge survey in international perspective](#)
 - [On three dimensional simulation of thermal behaviour of buildings](#)
 - [On visualization of heat and vapour diffusion stream in three dimensional models](#)
 - [The building envelope as thermal heat bridge](#)
 - [On the Storage of Heat in Building Components](#)
 - [Transient, multidimensional calculations by an example of components in contact with soil](#)
 - [Heat Transfer in Building Constructions in consideration of Heat Sources](#)
 - [A practical method for the assessment of heat bridges with a view to surface condensation](#)
 - [Appropriate responses to the complex problem of finding the cause for mould growth?](#)
 - [The concept of thermal conductance](#)
 - [Remarks to EN ISO 13786](#)
 - [Calculating the specific heat capacity of an equivalent component](#)
 - [The effective thermal heat capacity](#)
 - [Thermal conductivity and heat storage within planar components \(slabs\)](#)
 - [On the definition of the effective thermal heat capacity](#)
- [Tutorials and Cookbooks](#)
 - [Modelling and Evaluation Tutorial](#)
 - [Object of Analysis](#)
 - [Stages of Analysis](#)
 - [Two-dimensional Analysis](#)
 - [Generating the Model - Input](#)
 - [Completing the Tables](#)
 - [Completing the Component](#)
 - [Obtaining basic results](#)
 - [Graphic results](#)
 - [Comparing Results under Different Conditions](#)
 - [Three-dimensional Analysis](#)
 - [Generating the Model - Input](#)
 - [Completing the Model in Three Dimensions](#)
 - [Evaluation and Output in Three Dimensions](#)
 - [VAPOUR - Tutorial](#)
 - [Usage Hints Introductory Tutorial](#)
 - [Starting the application - main windows](#)
 - [Loading a component, primary windows](#)
 - [Calculation and Results](#)
 - [Evaluation of results](#)

- [Further instructions](#)
 - [Modelling by dynamic overlapping of elements](#)
 - [Various types of elements](#)
 - [The order of elements - overlapping effects](#)
- [Transiente Simulation - Tutorial](#)
- [Example images](#)
- [Example: Results report \(edited\)](#)
- [Example: Coupling Coefficients report \(edited\)](#)
- [FAQ - Questions & Answers](#)
- [Supplement](#)
 - [Install/1400x1050_AnTherm_Screen.png](#)
 - [Install/1024x768.png](#)
 - [Install/1280x1024.png](#)
 - [Install/1600x1200.png](#)
 - [Install/1600x1200_transparent.png](#)
 - [Forms/ElementBrowserForm/ElementBrowserForm.png](#)
 - [Forms/ElementBrowserForm/ElementBrowserForm_Context.png](#)
 - [Tutorials/tutor01/00_Start.MainForm.png](#)
 - [Forms/MainForm/MainForm.png](#)
 - [Forms/Elements2DForm/Elements2DForm.png](#)
 - [Forms/Elements23Form/Elements23Form.png](#)
 - [Forms/FineGridParamsForm/FineGridParamsForm.png](#)
 - [Forms/SolverParamsForm/SolverParamsForm.png](#)
 - [Forms/Result3DForm/Result3DForm.png](#)
 - [Forms/ApplicationSettingsForm/ApplicationSettingsForm.png](#)
 - [Forms/SolutionFolderCleanerForm/SolutionFolderCleanerForm.png](#)
 - [Die Lufttemperatur im unbeheizten Kellergeschoß; eine Fallstudie \(PDF\)](#)
 - [Heat losses from large buildings through the ground \(pdf\)](#)
 - [Methoden der Erfassung von Wärmebrücken im internationalen Vergleich \(PDF\)](#)
 - [Zur dreidimensionalen Simulation des thermischen Verhaltens von Gebäuden \(PDF\)](#)
 - [Die Gebäudehülle als Wärmebrücke \(PDF\)](#)
 - [Berechnungsgrundlagen zur Beschreibung mehrdimensional ablaufender Wasserdampf-Diffusionsvorgänge \(PDF\)](#)
 - [Berechnung äquivalenter Wärmeleitfähigkeiten für Hohlräume \(PDF\)](#)
 - [On calculation of \$\Psi\$ -values for building constructions in connection with ground \(PDF\)](#)
 - [On calculation of \$\Psi\$ -values \(Psi\) for a 3 space case: Special consideration: Unheated spaces \(PDF\)](#)
 - [Forms/RegisterUnderlayImage/RegisterUnderlayImage.png](#)
 - [On the Storage of Heat in Building Components](#)
 - [Heat Transfer in Building Constructions in consideration of Heat Sources](#)
 - [A practical method for the assessment of heat bridges with a view to surface condensation](#)
 - [Forms/EcoTechBaustoffeForm/EcoTechBaustoffeForm.png](#)
 - [Forms/TimelinesForm/TimelinesForm.png](#)
 - [Transient, multidimensional calculations by an example of components in contact with soil](#)
 - [Remarks to EN ISO 13786 \(2000\) \(PDF\)](#)
 - [Calculating the specific heat capacity of an equivalent component \(PDF\)](#)
 - [The concept of thermal conductance \(PDF\)](#)
 - [The effective thermal heat capacity \(PDF\)](#)
 - [Thermal conductivity and heat storage within planar components \(PDF\)](#)
 - [On the definition of the effective thermal heat capacity \(PDF\)](#)

Copyright

Information in this document is subject to change without notice and does not represent a commitment on the part of the vendor or its representatives.

No part of this document may be reproduced or transmitted in any form or by any means, electronic or mechanical, including photocopying, without the written permission of Tomasz Kornicki, Vienna A-1230.

Products named herein may be trademarks of their respective manufacturers and are hereby recognized. Trademarked names are used editorially, to the benefit of the trademark owner, with no intent to infringe on the trademark.

[Acknowledgments and Additional Terms of Use](#)

Furthermore the [License Terms and Conditions](#) and [General Business Terms and Conditions of Kornicki DL in EDV&IT](#) in their latest valid version are valid.

[Copyright: © Kornicki Dienstleistungen in EDV & IT](#)

License Terms and Conditions of Use

The Use of this Software Package is granted by terms and conditions of use shown below. Before opening the package or installing the Software you shall have read this contract carefully.

By installing the Software Package this contract becomes legally binding. The Right of Use, Transfer or Copying is restricted by this agreement.

The Software Package contains Data Processing Programs and the related Documentation. For both the term "Works" are used below.

The Licensor and other third parties own property right on these Works. Related to property rights of third parties the Licensor has sufficient rights to use. [Acknowledgments and Additional Terms of Use](#) apply also.

The Licensor grants to the acquirer (later called Licensee) the right to use the Works bound by following conditions:

1. Right of Use

- a. Works shall be used on or in combination with only one and single workplace or computer.
- b. If Works are being processed or linked with other Works, these other Works shall also be used on only one and single workplace computer.
- c. Unchanged, processed or linked Works shall be copied in the machine readable or printed form only for the purpose of securing the data or to process the Works. Thus 1a also applies for such copies.
- d. Any further or different use is not allowed.

2. Copyright Notice

Each part of Works contains a [Copyright Notice](#). Each Copy, each Processing and any Part of Works, which are linked to other Works must show this Notice also.

3. Transfer of the Right to Use

a. The Licensee can transfer his limited Right to Use, which result from 1. and 2, in full to a third party only with additional written permission from Kornicki and only if this third party provides a declaration of accepting the terms and conditions of this license contract. This Terms and Conditions shall be explicitly pointed out to the third party.

Kornicki will grant such a written permission, provided that the third party will oblige himself to update the Works to their newest available version on his expense and to cover any expenses of extending any Sublicenses to Full Licenses if applicable. The third party will also oblige himself to meet the costs of the transfer fee according to the then valid tariff list, payable to Kornicki.

Performing the transfer all Rights to Use granted to the Licensee are terminated, also including all Copies, Processing and Linkages. Shall those be not transferred to the third party, they must be irrevocably destroyed.

b. Transfer by the contract of any continuing obligation (e.g. rental, leasing) is prohibited.

4. Acquisition of extended version of the Software Package

Shall the Licensee extend, with these Works, an earlier version of Works or of Works linked to it, he is only granted the right to use this extended version. If nothing special has been separately agreed with the Licensor the Licensee is obliged to irrevocably destroy or delete all earlier versions and its copies, including the documentation, processing and linkages.

5. Licensor cannot be made liable for the correctness of any processing.

6. Additional terms for surrender of hardware dongles for the purpose of license activation

If the Licensee has opted to activate the license by the means of hardware dongles (instead of binding to the network adapter of one computer) the agreement extends as follows:

a. Hardware dongles shall be used only in connection with the use and activation of Works as set in 1. The licensee is entitled to install the Works on a maximum of 5 (five) computers but use the Works at one of them only (the one with the dongle installed).

b. The hardware dongles and data stored therein are property of the Licensor (The Owner) must not be changed, manipulated, simulated, copied, transferred or sold without the consent of the owner. Shall agreement be terminated for any reason the Licensee will, at his own expenses and without any additional notice, return hardware dongles to the owner immediately. At any time the owner is entitled to claim the return of dongles within 14 days at expense of the Licensee. At any time the owner is entitled to check the proper use of hardware dongles in the premises of the licensee or by electronic means.

Loss or damage to the dongles must be reported to the owner immediately. The owner is entitled to charge a fee to the Licensee for such replacement. After the payment of the fee by the Licensee the Owner will attempt to replace the dongle as soon as possible. If necessary, the entire stock of the dongles at Licensee's hands must be returned to the owner immediately.

c. With the handover of the dongles to the licensee or his representative or the dispatch to the transportation person, the risk goes to the licensee regardless of the fact who has paid for the shipment.

In case the Licensor is responsible for defect of the hardware dongle, the Licensor is authorized for his election for remedial or replacement. Obvious deficiencies must be complained within 14 days after receipt of the hardware dongle. Shall no complaints be made within 10 working days after the arrival of dongles at their destination, they shall be deemed approved. The guarantee expires at interference, repairs or repair attempts by the Licensee or any unauthorized third parties. The transfer of warranty claims is excluded. Replaced parts will also be our property. Same warranties apply to replaced or repaired parts. If the Licensor is unwilling or unable for replacement, or at least three remedial fail, the Licensee is entitled to rescind the contract. It is expressly stated that due to the wide variety of configurations and the constant developments in the peripherals the proper function of dongle modules cannot be assured in each every case. If nothing else has been explicitly agreed any further claims by the Licensee - regardless of what legal grounds - are excluded. Licensor cannot is not liable for any damages other than the dongle itself, especially not for lost profits or other property damage.

7. General Business Terms and Conditions

The [General Business Terms and Conditions of Kornicki DL in EDV&IT](#) in the latest version apply for the rest.

[Copyright: © Kornicki Dienstleistungen in EDV & IT](#)

Acknowledgments and additional Terms of Use

Acknowledgment WAEBRU:

(I)WAEBCOM is part of instationary WAEBRU only marginally related to WAEBRU Package. Interim integration into AnTherm "Codename Walter" granted by Dr.K.Krec/T.Kornicki.

DONGLE - Compliance with the CE / FCC regulations (CE)

The manufacturer of dongles has provided following statement:

This equipment has been tested for and approved to comply with the limits for Class B digital devices. The operation is subject to the following conditions:

1. The device may not cause harmful interference radiation,
2. The device must be able to process interference radiation, including such radiation which might lead to an undesired operation.

The product meets the limits set in EN55022 Class B, EN50081-1, EN50082-1 and EN55024.

A change to the product without express consent from the manufacturer, can cause the CE / FCC regulations are no longer being met. In such case the right to use the product by the Licensee extinguishes immediately.

Copyright

AnTherm® and all related works are [Copyright: © Kornicki Dienstleistungen in EDV & IT](#)

Licence terms and conditions of use

The use of this software package is bound by the [License Terms and Conditions of Use](#).

Validation according to EN ISO 10211:2007

The program package **AnTherm** fulfils all requirements raised in **EN ISO 10211:2007** for the simulation program to qualify it as **two- and three-dimensional, stationary precise method („Class A – Method“)**.

The EN ISO 10211 "*Thermal bridges in building construction — Heat flows and surface temperatures — Detailed calculations*" in its *appendix A* provides the *validation procedure* by which a "thermal bridges program" must conform to, to be qualified as "high precision method". **The application package AnTherm conforms in its current version to all respective criteria specified by EN ISO 10211:2007 for "high precision calculation methods" and thus it qualifies as "two- and three-dimensional steady-state high precision method (class A method)"**

Details of validation and benchmark results are put together [the calculation report of validation examples from the EN ISO 10211:2007 by the application package Antherm.](#)

- Reference case 1 (two-dimensional; temperature distribution within "half of a square column")
By the precision set within the standard (1 decimal place after comma) **AnTherm provides exactly concordant results**. By concerning uncertainties resulting from rounding **there is full concordance of results calculated by AnTherm with values calculated analytically**.
- Reference case 2 (two-dimensional; temperature distribution and heat flow within a roof construction)
This reference is considered shaky due to the temperature and heat flow calculation requirement within the junction made of a aluminium bracket nested within insulating material.
By concerning uncertainties resulting from rounding **there is full concordance of results calculated by AnTherm with values set in EN ISO 10211:2007**.
- Reference case 3 (three-dimensional; temperature distribution and heat flows through the wall-balcony junction) By concerning uncertainties resulting from rounding **there is full concordance of results calculated by AnTherm with values set in EN ISO 10211:2007**.
The difference between heat losses of interior spaces α and β and gains of the exterior (space α) calculated by *AnTherm* and those specified by EN ISO 10211:2007 are smaller than 0.11 %. The requirement set in EN ISO 10211, that differences of calculated heat flow values must not exceed 1%, **is met by AnTherm in full**.
- Reference case 4 (three-dimensional; temperature distribution and heat flows within a three dimensional thermal bridge consisting of an iron bar penetrating an insulation layer)
By concerning uncertainties resulting from rounding **there is full concordance of results calculated by AnTherm with values set in EN ISO 10211:2007**
The difference of 0.43% between the calculated heat flow and the value set by the standard is below the allowed tolerance of 1% as required by the reference case; The temperature calculated at the hottest point at exterior surface deviates by 0.003K which is below the tolerance of 0.005K set by the standard for that reference case; the criteria are **met in full by AnTherm**.

Read more here:

- [Details of validation can be found in the validation report. This report is located at the AnTherm's Web-Site.](#)

See also: [Validation according to EN ISO 10077-2:2003, Main menu Help->Example Load...](#)

Validation according to EN ISO 10077-2:2003

The program package **AnTherm** fulfils all requirements raised in **EN ISO 10077-2:2003** for the simulation program to qualify it as **standard method for calculation of heat flow through frames of windows, doors and shutters**.

The EN ISO 10077-2:2003 "*Thermal performance of windows, doors and shutters - Calculation of thermal transmittance - Part 2: Numerical method for frames*" in its *appendix D* provides the *validation procedure* by which a "thermal bridges program" must conform to, to be qualified as sufficient for the calculation of heat flow through frames conformant to the standards. **The application package AnTherm conforms in its current version to all respective criteria and assessments specified by EN ISO 10077-2:2003 and thus it can be used as "standard method for calculation of heat flow through frames of windows, doors and shutters"**

Detailed validation and benchmark results are put collectively into [the calculation report of validation examples from the EN ISO 10077-2:2003 with the application package Antherm](#).

- Reference cases 1 through 7 (thermal conductance L_f^{2D} and thermal transmittance U_f of frame profiles)
By concerning uncertainties resulting from rounding **there is full concordance of results calculated by AnTherm with values set in EN ISO 10077-2:2003**.
The differences between thermal conductance L_f^{2D} calculated by *AnTherm* to those specified in the EN ISO 10077:2003 stay below 1.1%. The requirement set in EN ISO 10077-2, that differences of calculated thermal conductance must not exceed 3.0%, **is met by AnTherm in full**.
The differences between thermal transmittance U_f calculated by *AnTherm* to those specified in the EN ISO 10077:2003 stay below 1.7%. The requirement set in EN ISO 10077-2, that differences of calculated thermal transmittance must not exceed 5.0%, **is met by AnTherm in full**.
- Reference cases 8 and 9 (thermal conductance L_f^{2D} and transmittance U of profiles)
By concerning uncertainties resulting from rounding **there is full concordance of results calculated by AnTherm with values set in EN ISO 10077-2:2003**.
The differences between thermal conductance L_f^{2D} calculated by *AnTherm* to those specified in the EN ISO 10077:2003 stay below 1.2%. The requirement set in EN ISO 10077-2, that differences of calculated thermal conductance must not exceed 3.0%, **is met by AnTherm in full**.
The differences between thermal transmittance U calculated by *AnTherm* to those specified in the EN ISO 10077:2003 stay below 0.5%. The requirement set in EN ISO 10077-2, that differences of calculated thermal transmittance must not exceed 5.0%, **is met by AnTherm in full**.
- Reference case 10 (thermal conductance L_f^{2D} and linear thermal transmittance Ψ of glazing, spacer and frame compound)
By concerning uncertainties resulting from rounding **there is full concordance of results calculated by AnTherm with values set in EN ISO 10077-2:2003**.
The difference between the thermal conductance L_f^{2D} calculated by *AnTherm* to the value specified in the EN ISO 10077:2003 stays below 0.2%. The requirement set in EN ISO 10077-2, that differences of calculated thermal conductance must not exceed 3.0%, **is met by AnTherm in full**.

The difference between the linear thermal transmittance Ψ calculated by *AnTherm* to the value specified in the EN ISO 10077:2003 stays below 2.4%. The requirement set in EN ISO 10077-2, that difference of calculated linear thermal transmittance must not exceed 5.0%, **is met by AnTherm in full**.

Read more here:

- 🔗 [Details of validation can be found in the validation report. This report is located at the AnTherm's Web-Site.](#)

See also: [Validation according to EN ISO 10211:2007, Main menu Help->Example Load...](#)

System requirements

Hardware requirements

- Compatible PC
- at least 512 MByte RAM (≥ 1 GByte for 3D-visualisation preferred)
- 200 MByte free space on disk
- Screen resolution at minimum of 1280x1024 (≥ 1280x1024 preferred) - [see below for further hint](#)
- Supporting 3D OpenGL graphics (minimum, optimum with 3D accelerated graphics board)
- B/W Printer A4 (minimum, colour printer preferred)
- Ethernet Network Adapter (for [license identification](#)) or the USB [license dongle](#) (purchased separately).
- Mouse with two buttons and scroll wheel (preferred)
- Internet connection (optional, eventually required during installation)
- Multiprocessor/Multicore-processors ([optional](#))
- Anaglyph Red/Cyan Glasses ([optional](#)), Red/Blue Glasses ([optional](#)), Crystal Eyes Glasses and Display ([optional](#)), Dresden Interlaced Display ([optional](#)), VRex Interlaced 3D projector ([optional](#))

The license of this program is permanently **bound to a special custom license dongle (USB) or to the MAC address of one physical Ethernet LAN card**. The license key will be computed based on four customer specified text lines (header to all reports) and the LAN address. After you complete the purchase process the vendor will contact you via e-mail to [obtain required data soon](#).

See also: [License terms and conditions](#)

Software requirements

- Windows 10, MS Windows 8 (32bit or 64bit), Windows 7 (32bit or 64bit), Windows Vista (32bit or 64bit), Windows XP Professional
- Microsoft® .NET 4.0 Runtime and .NET 3.5 Runtime installed (incl. V.2.0 and 3.0, will be automatically installed [as a free option directly from Microsoft](#) during the [Setup](#)).

Microsoft®NET4.0 Runtime is no longer required (from AnTherm Versin10 on)

- SAP Crystal Reports for Visual Studio 2010 runtime engine (will be automatically installed [as a free option directly from SAP](#) during the [Setup](#))
- A license to use the program *AnTherm*

SAP Crystal Reports is no longer required (from AnTherm Versin10 on)

The installation has been thoroughly tested with Windows XP Professional, Windows Vista and Windows 7 and 8. The execution of the program on systems like Windows XP Home, Windows NT SP6 or Windows 2000 SP4 shall be possible with slight limitations.

See also: [Installation of the program](#), [Installation of MS.NET Runtime](#), [License terms and conditions](#)

Memory requirements and license capacity

There is a 32-bit and a 64-bit version of AnTherm.

The 32-bit version has a limit of the memory that can be employed. It is by (theoretically) by 3 GByte. This limit raised by the infrastructure applies even when the 32-bit version of AnTherm is executed on a 64-bit system (e.g. Vista/Win7/Win8 64bit).

This **technical limit allows solving of systems up to 20.000.000 equations** and to receive resulting thermal heat conductance and numerical results of temperatures (160.000.000 nodes).

The limitation for the memory intensive **graphical evaluations is (currently) for models holding ca. 2.500.000 equation cells** (10.000.000 nodes).

Optimal display resolution

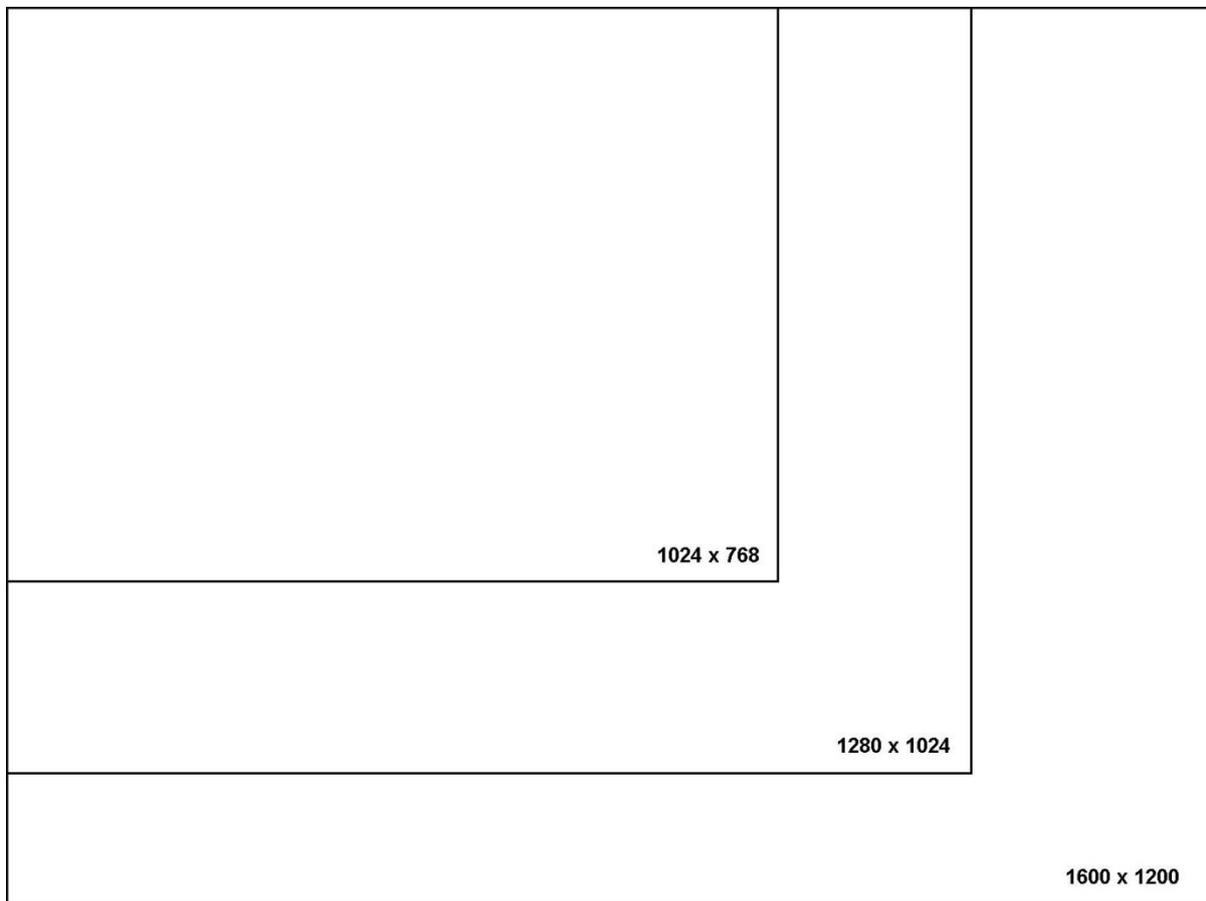
To comfortably work with AnTherm the display area available to the program shall be large enough.

With AnTherm you receive a highly interactive visualisation application. Modern visualisation technologies used by the program, with their roots in the world of supercomputers, raise certain demands and requirements towards capabilities of display devices used.

During long term experiences in use of the program it has proven, that the minimum requirement of 1280x1024 pixels display is the absolute minimum for comfortable interaction with the program - even if the whole display area is devoted only to AnTherm reaching some windows is only possible by scrolling. It is all but impossible to avoid overlapping of windows - even if the size of windows has been reduced up to the smallest limit, leading to very crowded working and output areas.

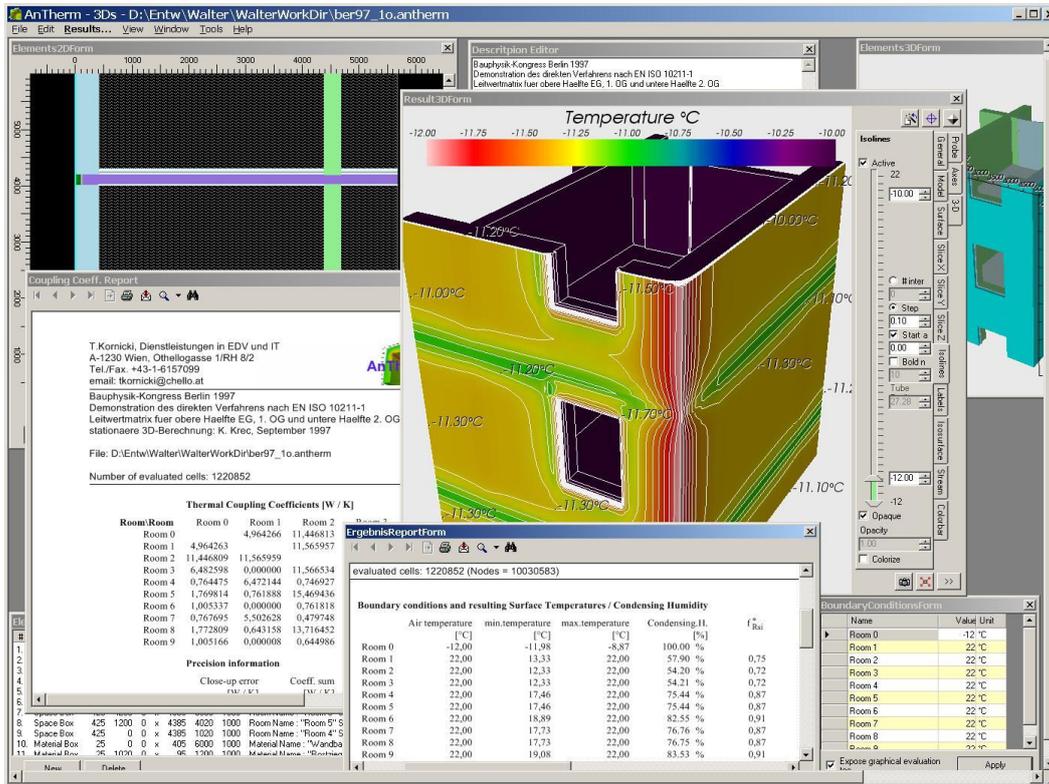
Optimal interaction conditions start at the resolution of 1280x1024. At resolution of 1600x1200 there is enough space on screen for AnTherm and for windows from other applications allowing the user to keep good control of the desktop.

Following graphics shall emphasize the relative sizes:



If you click on the picture above it will be shown in its original size on your screen (under certain conditions you will have to enable original size display in your browser).

The picture below shows AnTherm's window at 1400x1050 pixels. This size is sufficient to comfortably work with the software because it provides enough space for all application windows:



If you click on the picture above it will be shown in its original size on your screen (under certain conditions you will have to enable original size display in your browser).

Therefore we ask you to consider these minimum requirements when choosing equipment for your workstation.

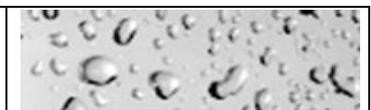
Short reference

- **Install** the software AnTherm® - by installing the software you explicitly declare your acceptance of [license term and conditions of use!](#) You will also have to install the [license file](#) .
- **Start** Antherm via Windows Start Menu **Start→Programme→AnTherm→AnTherm** . To check if installation was successful please select [Help→About...](#)
- Press **F1 key** to show the [application help](#).
- If needed you can change the [user interface language](#) to English (Englisch) or German (Deutsch)!
- Please consider also the additional dependencies of [country specific localisation](#) setting and the keyboard chosen. After changing these settings you will have to restart AnTherm to apply the changes.
- Open and load some example project via the menu [Help→Example Load...](#) . Follow instructions contained in [tutorials](#) and learn quickly how to effectively use AnTherm.
- Create your own, new project by selecting the menu [File→New→xxx Project](#)
- Reveal most important [data input/entry windows](#) by selecting the menu [View→Data Input & Entry→Primay Edit Windows](#)
- If you make a mistaken edit on your data revert your actions via [Undo \(Ctrl-Z\)](#) and [Redo \(Ctrl-Y\)](#) actions.
- Periodically save your current input data by using the menu [File→Save...](#)
- Request [calculation results](#) by selecting the menu [Result...](#) or depress the **F5 key** - after the calculation completes the [thermal conductance matrix](#) and calculation precision information will be shown.
- Define [boundary conditions](#) (temperatures, power densities) and apply them. The program will find [coldest surface points and respactive relative dew points together with fRsi values](#).
- Create [graphical evaluations in 3D](#) (surfaces, slices, colour maps, isolines, streamlines, iso-surfaces and many, many more) and [export](#) them to other applications for further processing.
- Finish and exit the program by selecting the menu [File→Exit](#).

Main menu:

- [File→New→2D Project](#): new 2D Model
- [File→New→3D Project](#): new 3D Model
- [File→New→Layered 3D Project](#): neues 3D Layered Model
- [File→Load](#): Open and load a AnTherm project file (also [drag&drop of the file over the main window](#))
- [File→Save/Save As...](#): saving current project data into a AnTherm project file
- [File→Import→Waebru.?BT](#) : import date from WAEBRU component file
- [File→Import→aCad.DXF](#): [import](#) data from specially prepared DXF file
- [File→Export→Conductance Matrix...](#): save the conductance matrix into a CSV file for further processing
- [File→Export→Report...](#): save a report into any of XLS, DOC, PDF, RTF compatible files
- [File→Exit](#): Finish and exit the application
- [Edit→Settings](#): [application settings](#)
- [Results...](#) (or the **F5 key**): [save](#) project data, execute calculation, entry of [boundary conditions](#), display of [result and evaluation windows](#)
- [View→Data Input & Entry→...Input windows](#): [input and data entry/modification windows](#)
- [View→Evaluation & Reports→...result and evaluation windows](#): various result and evaluation windows
- [Window→...](#): window navigation (MS-Windows specific)
- [Help→...various help](#): [application](#) information and application documentation
Pressing **F1 key** shows application help also.

VAPOUR-option: *Analysis of multidimensional vapour diffusion is only possible with an active VAPOUR-Option of the program*



[Input and data entry windows: Description, Element Editor, Materials, Surfaces, Element browser, Layers, Elements 2D, Snap grid, Translate, Rotate, Mirror, Stretch](#)

[The context menu of element processing](#) (by pressing the right mouse button over [Elements 2D, Element Editor, Element browser](#)):

- Edit Element: Show the [Element editing window](#).
- Add→Append/After/Before Selected: Creates and adds a new element
- [Duplicate](#): Creates and adds an exact copy of the selected element.
- [Move \(Order\)→Relegate/Promote](#): [reorders](#) the position of selected element(s) within the element's list
- [Select→By Group/Type/Name/All](#): [Selection](#) of elements
- [Copy, Cut, Paste](#): [Transfer](#) elements to/from clipboard.
- [Undo \(Ctrl-Z\)](#) and [Redo \(Ctrl-Y\)](#) your selection and editing actions

Element browser window (use functions of the [context menu of element processing](#) - by pressing the right mouse button over this window):

- Click in the list: Element selection. Selected element is shown in [Elements 2D](#) and [Element Editor](#).
- Double-click in the list: [Element Editor](#) will be shown.
- Right mouse button: show the [context menu of element processing](#).

Element Editor: (use functions of the [context menu of element processing](#) - by pressing the right mouse button over this window - and a double-click over material or surface data will reveal [Materials/Surfaces](#) lists):

- Use the TAB or Shift-TAB key to confirm data entered into the input field and to move the input focus between the fields and controls!
- Use arrow keys when the [element type](#) input field is focused!
[Empty Box](#) – excavating element. Elements „underneath“, i.e. listed earlier in the element list are overlapped with "Nothing".
[Material Box](#) – a material cell has material properties, like conductance, assigned to it.
[Space Box](#) – represents Space. A Space cell is used to assign "heat transfer coefficient" to the surface of the construction and then, during evaluation, to assign air temperature as the boundary condition for the space.
[Power Box](#) – the Power-Box frame. Material elements „underneath“, i.e. listed earlier in the element list will be defined as power sources (in addition to its material property). These areas are then assigned power boundary condition during the evaluation.
- Names of Spaces and Power Sources will be used to create the list of available [boundary conditions](#) during evaluation.
- A double-click over material or surface data reveals [Materials/Surfaces](#) lists
- Right mouse button: reveals the [context menu of element processing](#).
- Use the [ENTER key while the button "Duplicate"](#) is focused!

Windows of [Materials](#) und [Surfaces](#) are used to prepare of the later needed (and reusable) properties of [Materials-](#) or [Surfaces](#):

- A double-click transfer the selected property onto ([selected](#)) element(s)
- Right mouse button: Shows [Material-](#) or [Surface](#) properties and the Insert/Delete functions.

Elements 2D Windows (use functions of the [context menu of element processing](#) - by pressing the right mouse button over this window):

- Dragging with the mouse: pans the [view](#)
- Turning the mouse wheel: scales (zooms) the [view](#)
- Click onto an element's rectangle: [element selection](#) (single)
- Double-click onto an element's rectangle: the [Element Editor](#) is shown.
- SHIFT key held while dragging with the mouse: [Element selection with a lasso](#).
- Dragging the [Edit mark](#): Changes the coordinates of the element according [Snap-To-Grid settings](#).

Layer browser window: [selection](#) and editing of a [Layer](#) in a [Layered 3D Project](#)

- New/Delete: insert or delete a [Layer](#)
- Move Up/Down: moves the selected layer within the [list](#) of layers
- Click right mouse button: shows the [context menu of layer editing functions](#)
- Click left mouse button selection of a layer from the list of layers. [Element browser](#) and [Element 2D window](#) will show elements of this selected layer.
- Double-Click left mouse button: show [Layer editing window](#).

Elements 3D window: Is used for continuous visual verification during creation of 3D models.

- Drag with the mouse: rotates and rolls the view following the [Track-Ball principle](#).
with CTRL-key held roll around the Z axis.
with SHIFT-key held scales the view.
- Fit: will show the whole construction in the window.
- Brightening with [More Light](#), Showing as [Solid or as wire frame model](#)
- [Transferring the view via clipboard](#) to another application for further processing.

Boundary conditions window: Is used to enter boundary conditions for evaluation (e.g. temperatures). "Apply" start the superposition and the [evaluation](#).

Reports: [Data Entry Report](#), [Modelling Report](#), [Result Report](#), [Coupling Coefficients Report](#), [Psi-Value Determination:](#)
Textual output of results

- Menu File→[Print](#): prints the report
- Menu File→[Export](#)→[Report](#): export/save the report as Acrobat PDF, Excel XLS, Word DOC
- [File](#)→[Export](#)→[Coupling Coeff. Matrix...](#): save the matrix of thermal coupling coefficients to a CSV file
- [Controls on the Toolbar](#): [Zoom](#) the display, print, export etc.

Results 3D window: 3D visualisation of temperature distribution, heat stream density, heat stream and surface humidity.

- Drag with the mouse: rotates and rolls the view following the [Track-Ball principle](#).
with CTRL-key held roll around the Z axis.
with SHIFT-key held scales the view.
- Fit: will show the whole construction in the window.
- Evaluation views: [model](#), [isolines / isotherms](#), [iso-surface](#), [probe](#), [slice X,Y,Z](#), [streamlines](#), [vector arrows](#), [profile diagram](#)
- [Secondary functions](#): [Active \(scalar\) function](#) and [active vector field](#): temperature °C, heat stream density W/m², relative surface humidity %, saturated vapour pressure hPa, partial vapour pressure hPa, vapour pressure difference hPa, relative interstitial core humidity %, vapour stream density mg/m²h, vector field of heat stream W/m², vector field of vapour stream mg/m²h.
- Collecting and [interactively moving](#) or picking [points of interest](#)
- [Colouring](#) by given [colour map](#), [adjusted value range](#), and [opacity/transparency](#), brightening with [More Light](#)
- Representation as [solid or wire frame model](#)
- Setting and quick restoring of [3D results evaluation parameters](#) and [isometrics](#)
- [Transferring the view via clipboard](#) to another application for further processing.

Extensive [introduction](#) and [introductory tutorials](#), collections of [pictures](#).

[Validation according to EN ISO 10211:2007](#), [Validation according to EN ISO 10077-2:2003](#)

Further study of [theory and background](#) and [primary concepts](#) together with related [literature](#).

Please accept [licence terms and conditions](#) including requirements related to copyright marks on printouts.

Installation of the software

Important note: By installing the software [license terms and conditions](#) are in force.

Please check prior to the installation if the [requirements for proper execution of this application](#) are sufficiently met.

Important: You will need administrative rights (user logon) which permit you to install software.

Important: Internet connection might be required if prerequisite components must be downloaded automatically for the installation to complete.

To install the program please start the installer **Setup.EXE** and follow the instructions displayed on the screen.

Advice: If the software shall be run by different user than the one who installed it, then the **installation for all users** must be chosen (the choice has to be done within the dialog while selecting the installation folder)! If you wish to make the program available to all users of the workstation then please choose the option "All Users" at the related stage of the installation.

Important: The end user (or system administrator) **MUST use Setup.EXE** to download and install all required prerequisites. Running the Setup.EXE supplied with the package automates all required download and installation steps. Re-Running the Setup.EXE will repair the installation and install prerequisites if needed too.

After the installation completes successfully you can [start the program](#) and begin with your work. Learning the software is supported by [tutorials \(cook books\)](#).

Important: After completing the installation you shall activate the license with your personalized [license file AnTherm.HID](#) - see [License activation procedure](#) .

The installer will perform following steps automatically:

- The installer Setup.EXE checks and installs additional runtime components prerequisite for the execution (e.g. ".NET Runtime", "Crystal Reports Runtime") and copies all application files into the installation folder.
- Note: During the installation of prerequisites the user will be asked to confirm each installed package and accept licence terms of the each.
- Typically the installation folder is placed under "Program Files" folder of Windows (C:\Program Files\tkornicki\AnTherm). Example projects, documentation etc. is also copied to this folder.
- During the installation the installer will add to the user's "Start Menu" (or to the start menu of all users) the menu item "Antherm" and submenus allowing the user to [start the program](#) an documentation browser.
- Files with the extension ".antherm" will be associated as "[AnTherm project](#)". Later, a double click on the AnTherm project file [will automatically start](#) AnTherm.

Important: It is possible that the installation script will ask you to install [Microsoft® .NET runtime components](#) also. This will only happen if these components have not been installed yet on your system. In the absence of the required version of MS .NET Runtime it is neither possible to install nor to execute the program. In such case please install [runtime components](#) by following the instructions displayed. When done restart the installation of AnTherm. Installing [Microsoft® .NET runtime components](#) is [required](#) even if higher version of .NET components is already installed.

Important: If you received the installation package **by downloading it, then make a backup copy of it on some additional media** to have it available if needed later.

Uninstalling the application

To uninstall AnTherm use the System Control panel (Add/Remove Programs / Software).

Note: Prerequisite [runtime components](#) installed can be removed via System Control panel only (Add/Remove Programs / Software). Uninstalling AnTherm will not automatically uninstall prerequisites which might be required for other applications also.

See also: [System requirements](#), [Installation of MS.NET Runtime](#), [Installation of Crystal Reports Runtime](#), [License file AnTherm.HID](#), [License terms and conditions](#), [Country specific localization](#)

Installation of MS .NET Runtime and further prerequisite components

Running the Setup.EXE supplied with the [installation package](#) automates all required download and installation steps. Therefore there shall be no need for the user to bother about it manually.

The explanation below gives you more background information in the case you decide to install prerequisites manually and under your full control.

Remark: Directly running the AnTherm...MSI script (instead of Setup.EXE) will not install nor check most of its prerequisites thus the application will eventually fail in the case one or more prerequisite components are missing. Example 1: If prerequisite component of [Crystal Reports](#) is not installed showing any Report within AnTherm will lead to an Exception: "Could not load file or assembly 'CrystalDecisions.Windows.Forms, Version=10.5.xxx.... The system cannot find the file specified."

Example 2: Running AnTherm...MSI installer directly will complain if .NET3.5 is not installed yet and offer manual installation from some Microsoft site - internet browser will open showing several download and installation options the user has to choose on his own.

Installation instructions for Microsoft® .NET Framework Version 3.5 and Version 4.0

Links shown below will forward you to the download area and to detailed installation instructions of Microsoft® .NET Framework Version 3.5 and 4.0 (valid at the time of writing this section).

Please note, that you must be logged on with an account of the user granted installation permissions.

The language version shall be the same as of your operating system:

- Original source of [Microsoft related to .NET 4.0 Redistributable Packages](#) (web bootstrapper)
- or try [Microsoft .NET Framework 4 \(Standalone Installer\)](#) for offline installations (or [this link alternatively](#))
and
- Original source of [Microsoft related to .NET 3.5 Redistributable Packages](#) (web bootstrapper)
- or try [Microsoft .NET Framework 3.5 Service pack 1 \(Full Package\)](#) for offline installation

Alternatively you can choose [Windows Update](#) (windows menu Start → Windows Update) and then in the group "Custom Install/Optional Updates" choose .NET 3.5 and .NET4 installation.

Important: Even if .NET4 is already installed the .NET2 (included in .NET3.5) must be also installed.

Important: Microsoft® NET 4.0 Runtime is no longer required (from AnTherm Versin10 on)

Installation instructions for SAP Crystal Reports (32bit /64bit)

Links shown below will forward you to the download area and to detailed installation instructions of SAP Crystal Reports for Visual Studio 2010 runtime engine x86 (valid at the time of writing this section).

Please note, that you must be logged on with an account of the user granted installation permissions.

The language version shall be the same as of your operating system:

Original source from [SAP Crystal Reports for Visual Studio 2010 runtime engine](#)

- "START YOUR FREE DOWNLOAD", find "SAP Crystal Reports **runtime engine** for .NET Framework 4 (32-bit)"
- Download the file **CRforVS_redist_install_32bit_13_0_1.zip** .

Important: For the 32-bit version of AnTherm the 32-bit version of the Crystal Reports runtime has to be installed. For the 64-bit version of AnTherm the 64-bit version of the Crystal Reports runtime must be installed.

Important: SAP Crystal Reports is no longer required (from AnTherm Versin10 on)

See also: [System requirements](#), [Installation of the software](#)

The license file AnTherm.HID

All functions of the application have to be activated by applying the [license activation procedure](#). As long as the license has not been applied and licensing procedure not fully executed, AnTherm will immediately change to [demonstrational mode](#). In this mode many of applications' functions are not available to the user.

To fully activate all licensed program functions the customer specific license file *AnTherm.HID* has to be copied into the installation folder or the [alternative location of the license file AnTherm.HID](#) must be set within application settings.

If the license file is missing, or if the license term has expired, the way the program works will be strongly restricted and many elements will become not usable.

The information about the currently installed license and its validity can be checked in the [About window](#) which can be reached from the [menu Help->About...](#) .

License activation procedure

If the activation of the end user license has not been executed yet, one has to apply explicitly for such activation. The vendor shall contact you by e-Mail shortly after the purchase of the installation package.

The license of the program is made non-transferable and **bound to the physical hardware identification** (dedicated [hardware dongle](#), [a physical Ethernet LAN network adapter card](#), etc.).

From the data contained in

- four lines of [customer header \(heading all reports\)](#) and
- data on the [hardware dongle](#) or other [physical computer hardware identifiers](#) available for licensing

a license key is being derived.

Provided that data you will receive your personalized license file **AnTherm.HID** sent to you by e-Mail.

Extract the file from the archive received if required and:

- copy the license file without any changes into any user folder, public folder or onto network drive
- (you shall choose any folder writing to which no special administrative credentials are required comfortably) and
- enter that [location into the extended application setting "Alternative license file AnTherm.HID"](#) or ["Alternative List of license files AnTherm.HID"](#),
and
- restart the application.

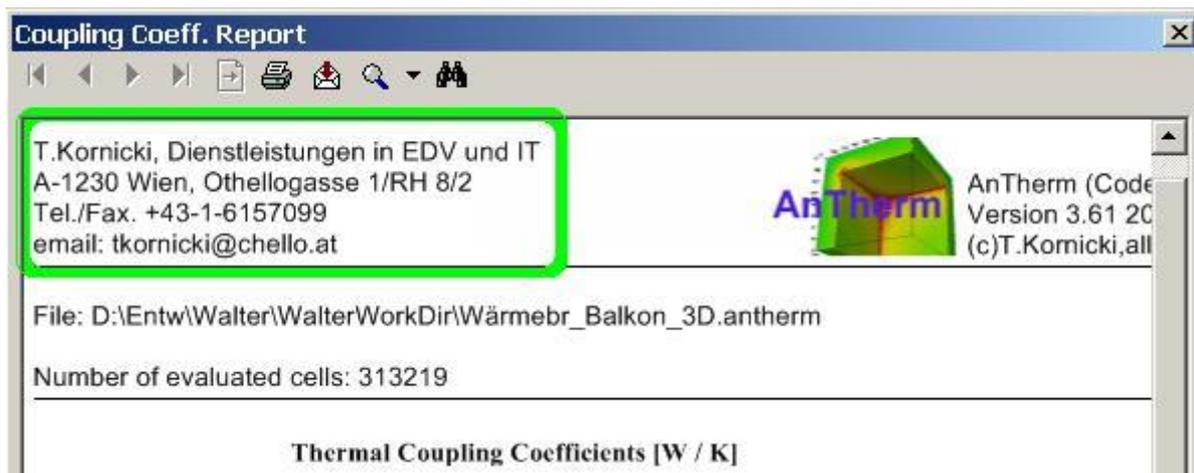
or (required only for rare very special cases):

- copy that file without any changes into the [installation folder](#) of this application (typically C:\Programs\tkornicki\AnTherm\)
- Important: Use this method only if the virtualisation of the installation folder (standard from Windows Vista on) has been disabled by acquiring proper administrative credentials prior to executing the copy command.

Important: We advise you to create **a backup copy of your personalized license file AnTherm.HID** to be able to restore it when required later.

Customer specific data (customer header)

Customer specific data of four text lines are printed in the heading of all reports



In accordance with [license terms and conditions of use](#) the licensee is not allowed to hide, remove or change any of vendor and user data shown in result reports.

Exact content of four available lines of text can be defined by the customer. This allows placing of company name, address and other information on heading of all reports. One shall only keep these lines short to allow enough space for the optional [customers' logo](#) placed in the middle of each heading - 50 to 60 characters on each line shall be sufficient in most cases.

The data, prepared in the way shown below, shall be made available by e-Mail to the vendor:

Customer data (maximum of 4 lines, each by max of 60 characters long):

Line 1:

Line 2:

Line 3:

Line 4:

Hardware identification:

Data provided shall identify the licensee truthfully and comprehensibly. The end user agrees that these data will be displayed in informational windows of the application and on all outputs produced by using this software. In accordance with license terms and conditions of use the licensee is not allowed to hide, remove or change any of vendor and user data shown in result reports.

Activating the license with a Hardware-Dongle

The activation with a hardware dongle adds mobility of the license between multiple computers - please verify the [license terms](#) on regulation of the ownership and the usage rights on one computer only.

DONGLE-option: replaces the binding of the license to one specific computer with the binding of the license to the mobile hardware, the USB-Dongle, allowing alternating use of one license on multiple computers.



Remark: Hardware dongle activation is an additional option which shall be purchased in extension to the license.

The activation of the license is combined with the data of the hardware dongle and stored in the license file. The application is able to execute only if the applicable hardware dongle, which is specified in the personalized license file, is plugged into the USB port of the computer.

Before you execute the application please plug the hardware dongle into any USB port of the computer on which the software and the associated license file are installed. There is no need to install any additional software (e.g. device drivers) to use the hardware dongle with AnTherm.

If you'd like to move the hardware dongle to some other computer please save your application data at first, then close the application and, after this, unplug the hardware dongle from the computer finally.

Remark: Never unplug the hardware dongle while the application is running. This might lead to unpredictable behaviour of the application or of the whole system or even damage the hardware dongle itself.

Remark: Protect the hardware dongle against loss or damage to avoid eventual blocking fine and repair or replacement costs.

Activating the license bound to physical computer hardware

Activating the license by binding it to physical computer hardware identifier validates the license for this one specific device - please verify the [license terms](#) on regulation of the ownership and the usage rights on one computer only.

The activation of the license is combined with the [data created out of the About window](#) and stored in the license file. The application is able to execute only on the computer, which is specified in the personalized license file. In the case of computer failure or replacement the license activation procedure must be repeated - incurring additional blocking, change and maintenance fees.

Please pass all *physical computer hardware identifiers* available for licensing directly out of the [About window](#) via e-Mail to the vendor. See: [About window](#).

Manual failure procedure

In rare cases a fallback to manual determination of device identification must be executed. Shall all the data passed fail to be available for licensing procedure the manual fallback identification based on physical MAC addresses of all Ethernet network adapters installed in the computer will be used. The physical *MAC addresses of one* (or even many) Ethernet network adapters installed in the computer are listed in the [top part of the about window](#), shown just after the licensing information and the list of active active license features:



Note: To readout the MAC address one can also use a helper application *adaptersAddresses.exe*. You will find *adaptersAddresses.exe* in the [installation folder](#) of AnTherm.

Note: To readout the MAC address the system command "*ipconfig /all*" can be used also. Listing of MAC addresses shown with this command can contain virtual network adapters however which are not suitable as license identification of the machine. Therefore please use the [output shown in the About window](#) or with the tool *adaptersAddresses.exe*.

```

is WinXP Professional or later
is WinXP or later
is Win98 or later
GUC-REALTEK Ethernet 10/100 PCI Adapter
is MIB_IF_TYPE_ETHERNET
Hardware address: 00-A8-B0-7C-F1-B0
  
```

Demonstrational mode

During execution of AnTherm in its *Demo-Mode* several application functions are not available. This demonstrational behaviour can be turned off by purchasing a permanent or a time limited license.

The license file controlling the demonstrational mode is located in subfolder "*DemoLicense*" of the installation folder. Properties of typical demonstrational license are restricted to following application features: *RESULTS*, *SOLVER*, *3DIM*, *30000CELLS*. Furthermore, a demonstrational license can be restricted to expire after some date.

- While in *Demo-Mode* the [About window](#) will periodically pop up. Each time the user must click it away manually.
- **The precision of results will always be worse than requested while in demonstrational mode.** The configuration of demonstrational mode targets short computational times at the expense of precision loss! The Demo-Mode's [termination condition](#) (the Delta) is set to 10^{-4} (0.0001); higher precision (required for conformity with valid norms) will be reached with terminal condition starting at and below 10^{-7} (0.0000001).
- For that reason physical interpretation of results produced with demonstrational version is not reasonable! The software will show a warning message if the user tries to create more precise results by reducing the termination condition:

The termination delta of the Solver-Engine has been reset to 0.0001. The demonstration license targets towards short computational time. The results are being computed faster but the precision might suffer.

You shall upgrade your license if you wish to obtain more precise results.
Please contact the manufacturer or reseller of this application to acquire full featured license.

- In *Demo-Mode* all pictures copied to the clipboard or saved to file [will be watermarked with a text „Demonstration“](#).
- In *Demo-Mode* all result reports are [watermarked with a text „Demonstration“](#).
- In *Demo-Mode* all 3D pictures will be [overlapped transparently with a text „Demonstration“](#); The license information is shown at the upper-left corner of the image also.
- In *Demo-Mode* all save and export functions of reports are not available.

Note: If the license is invalid (for example it has expired) intervals of About-window popup will become shorter and after 5 minutes the application will exit.

Note: Projects save a flag if executed in demo mode (*DemoModeApplied*). After license upgrade the termination delta will be reset to reasonable default value (*SolverParameters.DemoModeRevertToNormal*).

Features

Which application functions are covered and active within your license is defined within the license file by following feature entries:

- 2DIM – calculation of 2-dimensional constructions (2D)
- 3DIM – calculation of 3-dimensional constructions (multi-Layered or 3D)
(without 3D license there is no 3D editing/processing: disabled menus: File->New, File->Convert, File->Import, nor saving of 3D/3DS)
- RESULTS – executing the menu Results...
- EDURESULTS – executing the menu Results... with a [watermark "Educational"](#)
- DEMORESULTS – executing the menu Results... with a [watermark "Demonstration"](#)
- SOLVER – executing the Solver
- ROUNDNESS – the roundings
- 50000CELLS – executing the Solver up to 50.000 cells
- 100000CELLS – executing the Solver up to 100.000 cells
- 300000CELLS – executing the Solver up to 300.000 cells
- 1000000CELLS – executing the Solver up to 1.000.000 cells
- ANYCELLS – executing the Solver without cell limit
- VERSION number - required for execution
- NODEMOPUP – suppress periodic popup of about windows when in Demo-Mode.



VAPOUR-Option specific features:

- [DAMPF2DIM](#) - solving two dimensional vapour diffusion
- [DAMPF3DIM](#) - solving three dimensional vapour diffusion

HARMONIC/TRANSIENT-Option specific features:

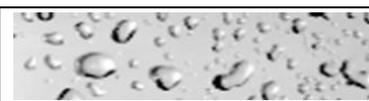
- [HARMONIC](#) - solving of the harmonic, periodic, transient heat transfer (e.g. phase shift, harmonic coupling coefficients, etc.).
- Allows solution for only 1 (first) harmonic for the [selected period](#) to be calculated. Only [harmonic thermal conductances](#) will be output. Other evaluations are executed with steady state solution (i.e. 0-th harmonic) and steady state boundary conditions (Results, ProbePoints, Results3D).
- [TRANSIENT](#) - in addition to [HARMONIC](#) allows solving and evaluating the transient problem with time dependant, variable, periodic boundary conditions with higher harmonics.
- Allows [free entry of higher number of harmonics](#) to Solver, [specification of Periodic Boundary Conditions](#) and executing Time-Based evaluations ([Timeline](#), [Results3D Animation](#)). The Results, ProbePoints and Results3D are calculated for [some chosen Time-Point \(SharedTime\)](#) also set within boundary conditions (and it "includes" HARMONIC by its nature!).

Additional Features:

- [MULTICORE](#) - performance improvement by executing solver distributed on multiple processors or processor cores, eg.:
 - Solving the equation system(s) - the Solver
 - Loading the solution vectors from files in solution folder
 - Combining steady and transient state solution vectors at specific boundary conditions (superposition)
 - Harmonic Fourier analysis and synthesis of periodic boundary conditions
 - Fourier synthesis of time dependant solutions
 - Creation of results data for 3D visualization (primary and secondary functions), timelines and animation
- [STEREO3DVIEW](#) - allows choice of [binocular 3D views](#) on supported devices
- 64 BIT- running the 64bit version

Vapour diffusion solution (VAPOUR-Option)

VAPOUR-option: Analysis of multidimensional vapour diffusion is only possible with an active VAPOUR-Option of the program..



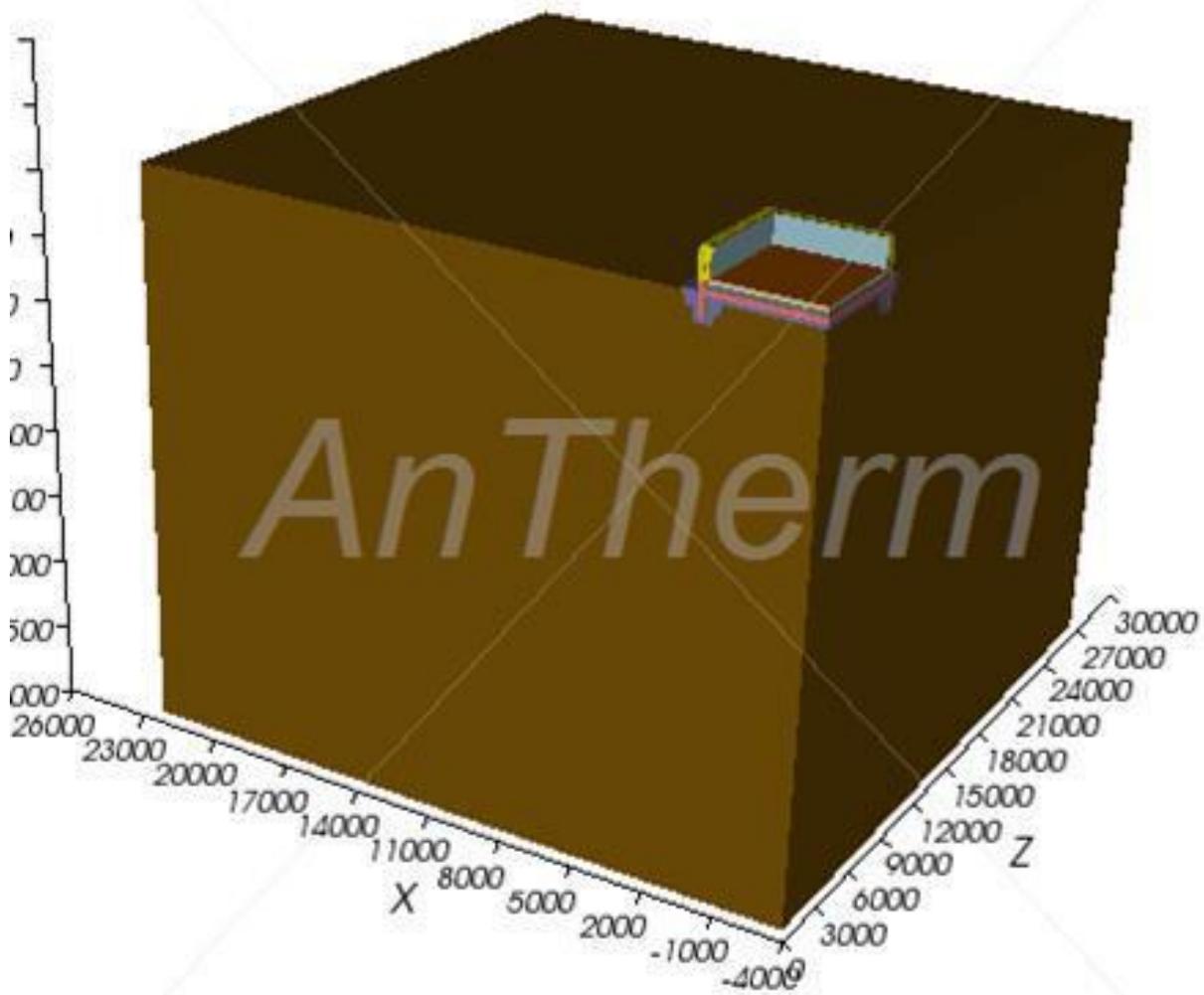
Computational description of vapour diffusion in building constructions can provide valuable hints for answering the questions if there is a risk of, potentially destructive, interstitial vapour condensation within the core of the construction or not (Interstitial Condensation Assessment).

Diffusion equations have the same structure as those describing heat transfer and can be solved analytically only for one dimensional vapour diffusion. In all higher dimensional cases numerical methods, like the one implemented in AnTherm, have to be used.

AnTherm calculates two- and three-dimensional distribution of vapour partial pressures in building constructions.

See also: [Installation of the program](#), [License terms and conditions of use](#), [System requirements](#), [License order form](#), [Application setting "Alternative license file AnTherm.HID"](#), ["Alternative List of license files AnTherm.HID"](#)

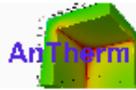
"Educational" watermark



Coupling Coeff. Report

T.Kornicki, Dienstleistungen in EDV und IT
 A-1230 Wien, Othellogasse 1/RH 8/2
 Tel./Fax. +43-1-6157099
 email: tkornicki@chello.at

06.07.2016

 AnTherm V.8.131 2016.06.14
 (c) Kornicki, all rights reserved

TUTORIAL - 2-dimensional:
 Reinforced concrete slab with masonry bearing wall on girder over
 carport (+ exterior insulation and electric heating cable assembly
 embedded in concrete floor topping).

File: T:\Benchmark\TUTOR2_orig_conv.anthem

Thermal Coupling Coefficients [W / m²K]

Space\Space	Room 0	Room 1
Room 0		0,920613
Room 1	0,920613	

Used Heat Transfer Coefficients

Room	Value	Surface
Room 0	Rs=0,04 m ² K/W	Ext. transfer coeff.
Room 1	Rs=0,1667 m ² K/W	Int. transfer coeff.

Distribution factors [1]

Pw\Space	Room 0	Room 1
PS 0	0,082675	0,917325

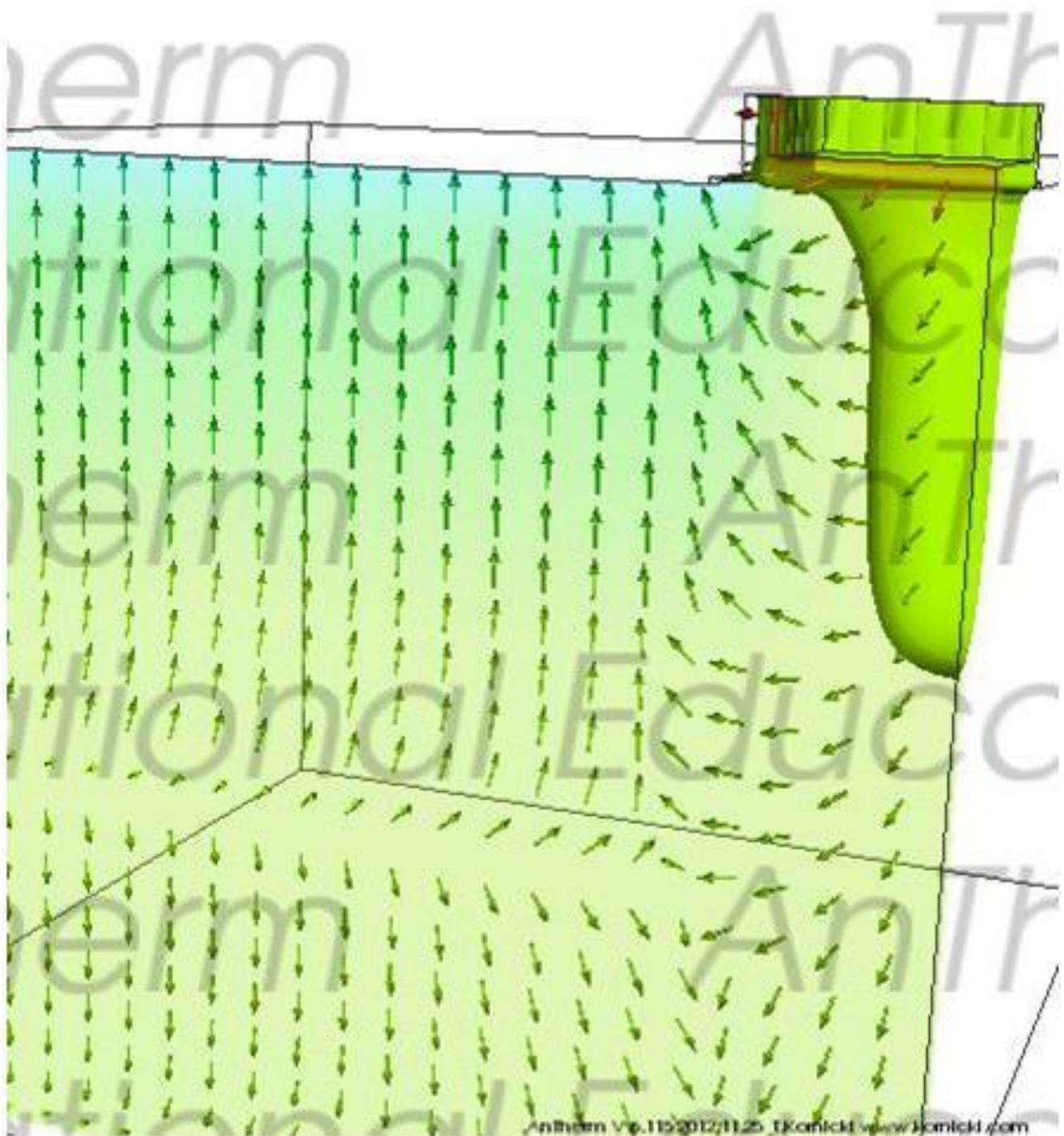
Precision information

Space	Close-up error [W / m ² K]	Coeff. sum [W / m ² K]	Relative close-up error
Room 0	4,52527e-009	0,920613	4,91550e-009
Room 1	-4,52527e-009	0,920613	-4,91550e-009

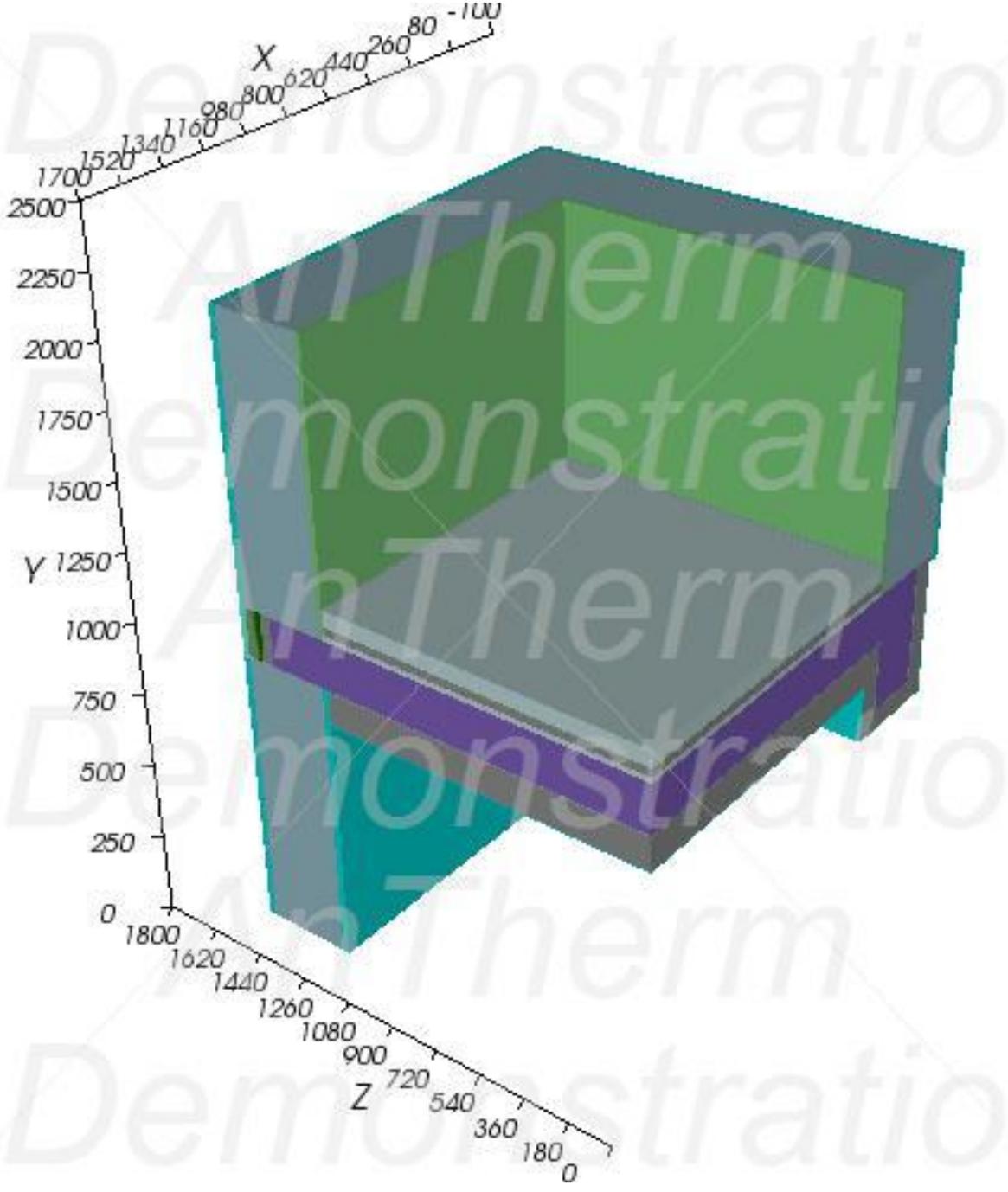
Power Source	Close-up error [1]	Total sum [1]	Relative close-up error
PS 0	1,35604e-008	1,000000	1,35604e-008

AnTherm - Heat & Vapour Transfer-Thermal Bridges- Walter,Udo,Klaus,Tomasz
 Copyright (c) 2003-2016 M.Kornicki Dienstleistungen in EDV & IT <http://www.kornicki.eu>

1



"Demonstration" watermark



Coupling Coeff. Report

1 / 1

M.Kornicki, Dienstleistungen in EDV und IT
 A-1230 Wien, Othellogasse 1/RH 8/2
 Tel./Fax. +43-1-6157099
 email: tkornicki@chello.at

AnTherm V.8.131 2016.06.14
 (c) Kornicki, all rights reserved

06.07.2016

TUTORIAL - 2-dimensional:
 Reinforced concrete slab with masonry bearing wall on girder over
 carport (+ exterior insulation and electric heating cable assembly
 embedded in concrete floor topping).

File: T:\Benchmark\TUTOR2_orig_conv.antherm

Thermal Coupling Coefficients [W / m²K]

Space\Space	Room 0	Room 1
Room 0		0,920613
Room 1	0,920613	

Used Heat Transfer Coefficients

Room	Value	Surface
Room 0	Rs=0,04 m²K/W	Ext. transfer coeff.
Room 1	Rs=0,1667 m²K/W	Int. transfer coeff.

Distribution factors [1]

Pw\Space	Room 0	Room 1
PS 0	0,082675	0,917325

Precision information

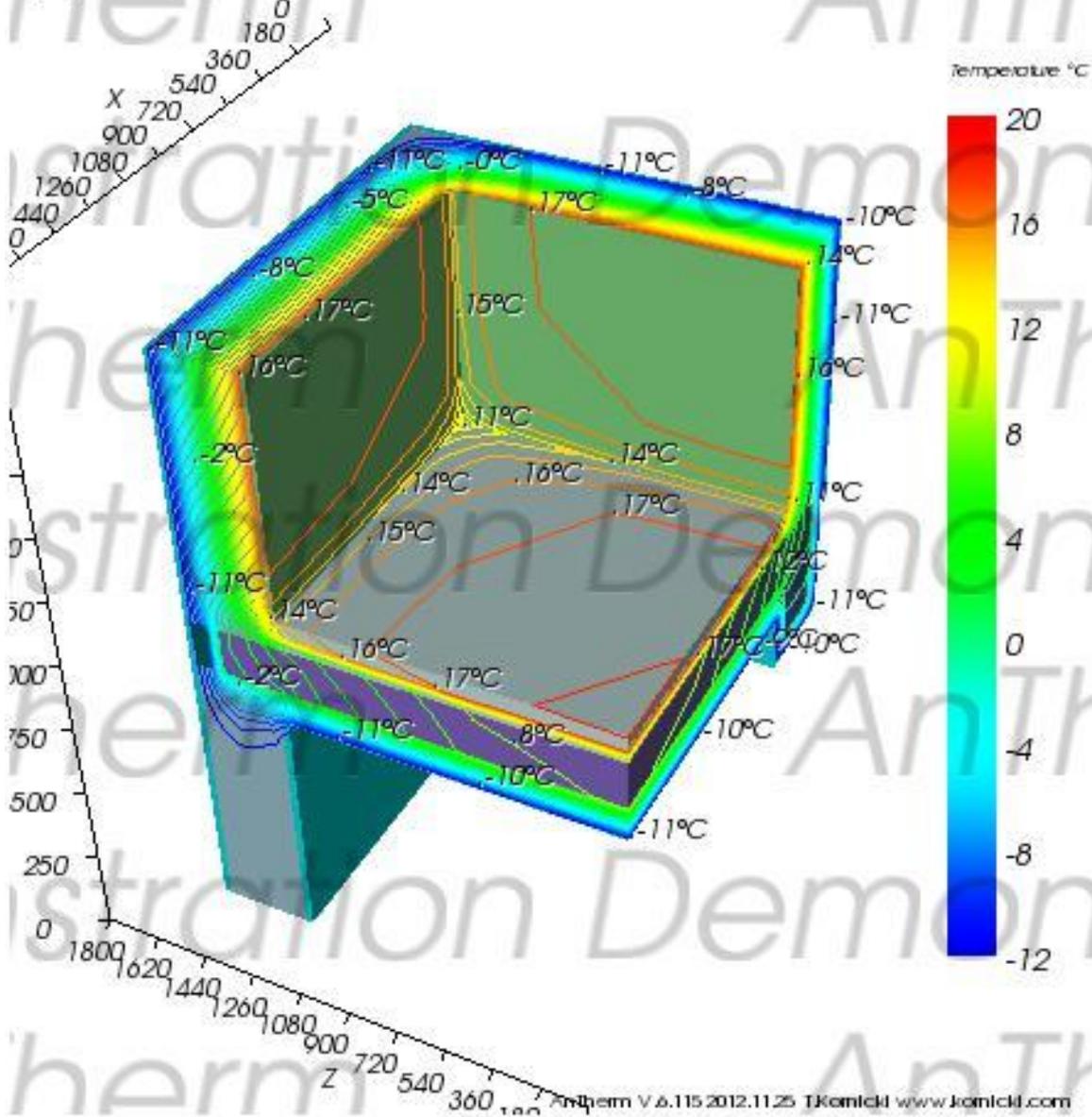
Space	Close-up error [W / m²K]	Coeff. sum [W / m²K]	Relative close-up error
Room 0	4,52527e-009	0,920613	4,91550e-009
Room 1	-4,52527e-009	0,920613	-4,91550e-009

Power Source	Close-up error [1]	Total sum [1]	Relative close-up error
PS 0	1,35604e-008	1,000000	1,35604e-008

AnTherm - Heat & Vapour Transfer-Thermal Bridges- Walter,Udo,Klaus,Tomasz
 Copyright (c) 2003-2016 M.Kornicki Dienstleistungen in EDV & IT <http://www.kornicki.eu>

1

Demonstration license - exclusive demonstration preview
Any production, duplication or disclosure is prohibited
Copyright (c) T. Kornicki, Dienstleistungen in EDV und IT
<http://antherm.eu/> email: antherm@kornicki.com



Introduction

The program AnTherm® is used to calculate stationary (steady state, time independent) and dynamic (transient in time periodically) distribution of temperature and heat flux within building construction, especially to determine surface temperatures and thermal coupling coefficients. It is also used to determine [vapour diffusion](#) in building constructions. Extensive visualization support and extend the analysis.

For the dynamic, transient case of changing, harmonic, periodic boundary conditions AnTherm will calculate harmonic coupling coefficients.

In that simple cases in which the temperature within the building component is only dependant on one location coordinate, i.e. in the case of one dimensional heat transport, there is no need for specialized computer programs; the calculations can be easily completed by using well known, simple formulas.

If the temperature distribution depends on two Cartesian space coordinates then we have to do with two dimensional heat transport and [linear thermal bridges](#). The general, three dimensional case of heat transport shall be considered if the temperatures depend from three space coordinates.

The program AnTherm® allows calculation of temperature distribution for the two as well as for three dimensional heat transfer.

The program AnTherm® has been developed for model calculation of such building components of which the boundaries are parallel to coordinate planes of the orthogonal Cartesian coordinate system. Thanks to such limitation to orthogonal network the practical input can be quickly executed. Any construction set up of interconnected or overlapping ashlar and arbitrary materials can be simulated. The limitation to orthogonal structures allows relatively [finely grained](#) calculation of temperature distribution even on typical personal computer. Since the temperatures are calculated not only at cell centres, but also at sides, edges and corners of each cell, the simulation of say 20.000 cells provides 80.000 node temperatures in two dimension or even 160.000 nodes in three dimension very quickly. For 1.000.000 calculated cell there will be more than 8.000.000 temperature nodes calculated in 3D!

On one hand the temperature distribution depends on the geometry and physical properties of the component and, on the other, from air temperatures within spaces attached to the component (boundary conditions). There can be more than only two spaces attached to the component - this is the additional and major difference of two- or three dimensional cases compared to one dimensional calculation.

Computational description of [vapour diffusion](#) in building constructions can provide valuable hints for answering the questions if there is a risk of, potentially destructive, vapour condensation within the construction or not. The implementation provided in AnTherm shall be mainly used for answering the question if, and if so where within the construction, by given boundary conditions (air temperature and relative air humidity) moisture is produced.



The distinctive feature of AnTherm® is that the relatively laborious calculation of the temperature distribution, which is provisioned by [solving large system of linear equations](#), need not be repeated for each different set of boundary conditions. Anticipating that need AnTherm determines so called "base solution" during the calculation only once. These base solutions are then superposed to temperature distribution under requested boundary conditions leading to significant reduction of computational time when several variants of boundary conditions are of interest.

With regard to [theoretical background](#) reference should be made to the book *"WÄRMEBRÜCKEN"* (Heindl, Kreč, Panzhauser, Sigmund; Springer-Verlag Wien-New York).

See also: [Tutorials and Cookbooks](#), [Primary Concepts](#), [Theoretical background](#)

Two- and Three-Dimensional heat flow Patterns

The method of analysis most commonly practiced today when evaluating the thermal performance of building spaces, components, and assemblies is based on a simple, one-dimensional, constant flow model of heat conduction (i.e. the assumption of parallel heat flow for the calculation of U-values and areas). Such an assumption often leads not only to disappointing results in the thermal performance of realized construction projects, but also to costly consequences due to

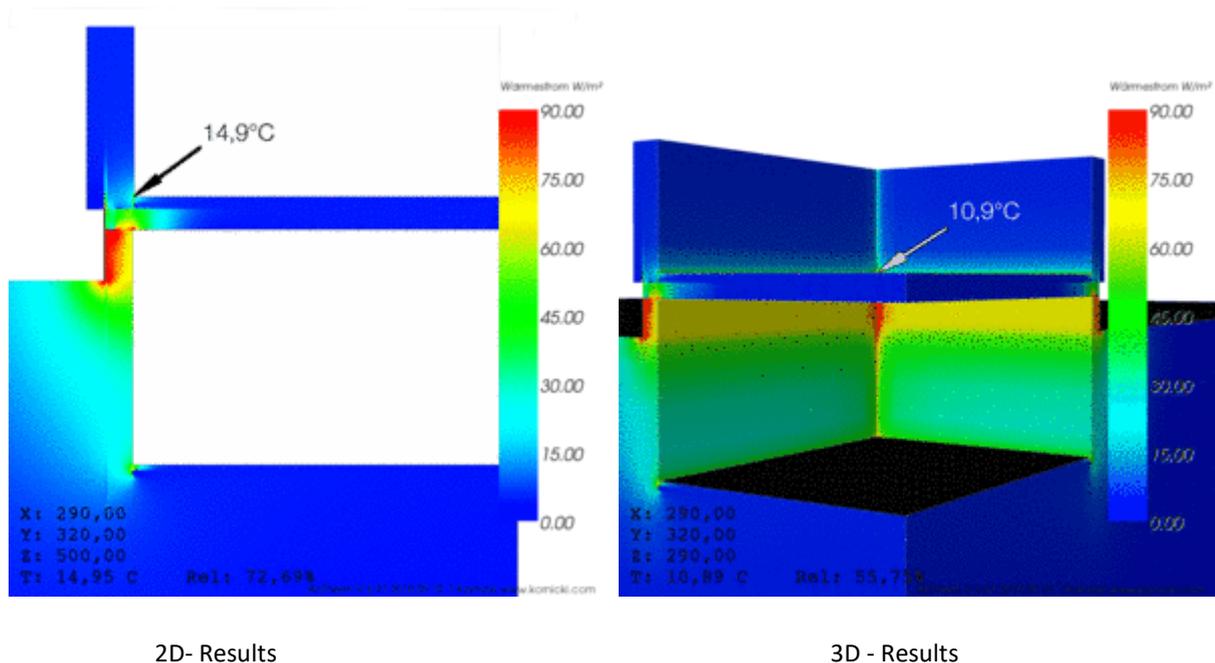
- unforeseeably high energy consumption for heating, as well as
- damage caused by surface and core condensation and moisture

These potentially negative consequences of over-simplification, inherent to the assumption of one-dimensionality, are becoming increasingly critical in today's trend towards highly insulated building structures. If the effects of [thermal bridges](#) are neglected, drastic errors in estimating heating requirements are bound to result, particularly when assessing energy efficient buildings.

Multi-dimensional (i.e. two- and three-dimensional) evaluations of thermally critical regions within a building assembly, especially those with thermal heat bridges, during early design phases can provide valuable preliminary information to support the decision-making process, thus leading to considerably more reliable design results.

Surface moisture due to condensation (typically occurring in such regions as floor-wall connections, window installations, etc.) as well as mould growth in humid environments can also be effectively prevented by means of multi-dimensional evaluation during planning and detail design.

See also: [Multidimensional Vapour Diffusion, European standards on thermal heat bridges](#)



As this example shows only the 3D evaluation returns the actual minimal surface temperature.

European Standards on Thermal Heat Bridges

The European standards "Thermal Bridges in building construction Heat flows and surface temperatures - Part 1: General calculation methods" (EN ISO 10211-1) pertaining to aspects of thermal performance of building constructions are already available since 1996. The EN ISO 10211:2007 "*Thermal bridges in building construction — Heat flows and surface temperatures — Detailed calculations*" shows further development of the standardisation. The standard EN ISO 6946 "Building Components or Building Elements - Calculation of Thermal Transmittance" is also available. The EN ISO 10077-2:2003 "*Thermal performance of windows, doors and shutters - Calculation of thermal transmittance - Part 2: Numerical method for frames*" shall be also considered.

The ramifications of the adoption of these European Standards became apparent: they describe the substance of essential tasks for designers, manufacturers, and builders, as well as their significance for quality control in the field of energy conservation.

The purpose of the standard EN ISO 10211 "Thermal Bridges - Calculation of Surface Temperatures and Heat flows" is explicitly stated as:

- the calculation of minimum (lowest) surface temperatures in order to assess the risk of surface condensation and
- the calculation of heat flows in order to predict overall heat loss from a building (for the constant, steady state flow case; *i.e. time independent temperature distribution*) and
- determination of linear and point thermal transmittance and surface temperature coefficients (of thermal bridges).

Furthermore, "[high precision methods](#)" of calculation are demanded, whereby precision criteria which must be satisfied by the method chosen are defined in the standard.

According to EN ISO 6946 "Building Components or Building Elements - Calculation of Thermal Transmittance", calculation of heat transfer coefficients of parallel plane surface building components shall be performed based, of course, on one-dimensional models. For such components, composed of surface parallel layers of thermally homogeneous or non-homogeneous materials, a well-known method of estimation (ISO-method) can be implemented in order to obtain a design thermal resistance - assuming the maximum relative error remains "negligible".

For all other heat flow patterns, i.e. for all cases of more than one dimension, in particular in the presence of thermal bridges, the standard EN ISO 10211 "Thermal Heat Bridges" requires the implementation of numerical methods of evaluation.

Further standards to be considered (exemplary):

- **ISO 7345** "Thermal insulation - Physical quantities and definitions"
- **ISO 10456** "Building materials and products - Hygrothermal properties - Tabulated design values and procedures for determining declared and design thermal values"
- **ISO 13370** "Thermal performance of buildings - Heat transfer via the ground - Calculation methods"
- **ISO 13786** "Thermal performance of building components - Dynamic thermal characteristics - Calculation methods"
- **ISO 13788** "Hygrothermal performance of building components and building elements - Internal surface temperature to avoid critical surface humidity interstitial condensation - Calculation methods"
- **ISO 13789** "Thermal performance of buildings - Transmission and ventilation heat transfer coefficient - Calculation method"
- **ISO 14683** "Thermal bridges in building construction - Linear thermal transmittance - Simplified methods and default values"
- etc.

A calculation program as instrument

In order to perform tasks imposed by European regulations in practice, the qualified planner, designer, or expert needs a calculation program which:

- meets the demands of the European Standards with respect to input data and more importantly output data requirements (evaluation results),
- is capable of executing calculation with [high precision](#), and is sufficiently convenient to the user in application (providing ease and clarity of operation, as well as speed).

AnTherm® possesses these instrumental characteristics

- AnTherm facilitates the generation of geometrical models by supporting a graphic input display of building structures, as well as by providing an entirely independent, fully automatic method of fine subdivision (which can also be influenced by the user through manual manipulation of parameters).
- It delivers complete input model documentation (geometry of material elements, thermal design values of materials, spaces, heat sources...) upon request.
- It allows the precision of numerical solutions to be influenced and controlled by the user (definition of calculation parameters).
- AnTherm provides results conformant to European standards:
 - generally applicable results in the form of **g-values** and **conductance matrices**. These conform to the **temperature weighting factors** and "**thermal coupling coefficients**" defined in the European Standards, including the required information on calculation precision,
 - specific results, applicable to particular air temperature conditions in spaces thermally coupled by the building components analyzed, in the form of **surface temperature minima and maxima** as well as respective **dewpoints**,
- It creates graphic plots and prints of **isotherms**, **surface temperatures** or **temperatures along an edge** (2- or 3-dim. models), as well as **heat flow diagrams** (not limited to 2-dim. models only, but **also in 3D**).
- AnTherm [meets all demands of the European Standards](#) with respect to input data and - more importantly - output data requirements (evaluation results) - validated according to EN ISO 10211: AnTherm has been qualified as a "Class A" tool – as for two- and three-dimensional, stationary precision method.

See also: [Validation according to EN ISO 10211:2007](#), [Validation according to EN ISO 10077-2:2003](#), [Structure of the application](#)

Structure of the program

AnTherm is a package of numerous programs branches which are grouped and coordinated centrally. Although the program is composed of several modules (program branches), it can be operated as though it were one single program. This is made possible by the unified user interface which independently ensures proper sequencing through the chain of programs while thoroughly managing the various objects generated by individual branches of AnTherm.

The program can be divided into following branches:

- the model input branch
- the calculation branch (or simulation branch)
- the evaluation branch (or output branch, results branch, analysis branch)

The individual program branches are briefly described here.

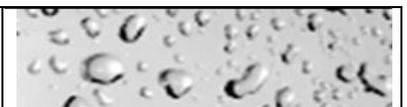
The input branch supports the generation of a calculation data suitable for describing the object under consideration. The first stage of input entails generating a model of the object with respect to its geometry and the thermal properties of its materials together with the specification of attached spaces and their heat transfer coefficients. Optionally one can identify areas as heat sources which will supplying specified power to the model. Finally the the level of detail for the equation grid has to be specified, i.e. the subdivision of the model into fine network of grid elements required for the calculation. These pieces (rectangles or ashlars) of the network are later called "cells".

Having completed the input branch without errors the calculation branch can start with determination of "base solutions" - the basis of results. On top of that network a large system of linear equations is set up and solved iteratively. The number of equations in the system is equal to the number of cells in the orthogonal grid structure which it describes; thus a very finely gridded structure (calculation geometry) can result in lengthy computation.

Final output data is obtained in the evaluation branch, during which the model under consideration (or rather, its characterisation through base solutions) is evaluated under specific conditions. The boundary conditions which must be specified here include air temperatures for the spaces/cases calculated. In the event that heat source elements have been included in the model, power values must be assigned as a condition to these cases as well. Herein lies the advantage of the "detour" over base solutions, since these need be calculated only once for a given model. The calculation time required for subsequent evaluation of specific temperature distributions under alternate boundary conditions is thus substantially reduced by employing the base solutions which have been determined in the calculation branch - instead of repeating the entire analysis for each set of conditions. Numeric data is printed here in the final form of evaluation results.

Graphic representations of a given temperature distribution are generated in a form suitable for further evaluation. Within the graphical output of temperature distribution - like surface temperatures - one can false colour the temperature distribution or emphasize it by displaying isotherms also. Also images of streamlines or stream flux can be shown emphasizing the heat flow through the considered construction, often helpful in the identification of weaknesses of the construction.

VAPOUR-option: Analysis of multidimensional vapour diffusion is only possible with an active VAPOUR-Option of the program



If material properties of vapour transport have been supplied also partial vapour density and vapour flux can be calculated and evaluated here.

The evaluation branch provides printing of numerical results. Graphical results can be easily transferred to other applications for further processing.

See also: [Input branch](#), [Gridding](#), [Calculation branch](#), [Evaluation branch](#)

The input branch

The input branch supports the generation of a calculation data suitable for describing the object under consideration. The first stage of input entails generating a model of the object with respect to its geometry and the thermal properties of its materials together with the specification of attached spaces and their heat transfer coefficients. Optionally one can identify areas as heat sources which will supply specified power to the model.

As with any numeric method of analysis, the application of AnTherm with respect to a concrete object requires a considerable abstraction of reality. For dependable results, building components must therefore be abstracted to conform to the geometric principles governing the numeric analysis performed by AnTherm.

Basics of the input process will be described in further sections for a two dimensional case. For the three dimensional case same basic rules apply also.

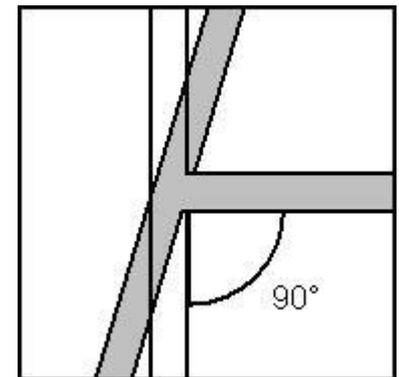
Abstracting the Geometry

First and foremost, AnTherm operates only within an orthogonal, Cartesian coordinate system. This limitation is due to the method of analysis, which, briefly put, first divides the model into a network of numerous cells ("gridding") to be mathematically evaluated. The mathematical description of these cells is a system of equations. The finer the grid, the more exact the evaluation results. However, a finer grid also, logically, increases the number of cells to be calculated, that is, the number of equations to be solved. This quantity is limited by the particular computer hardware configuration of the user. For orthogonally subdivided cells, a relatively simple system of equations suffices to describe each cell completely, thereby making it possible to evaluate a very finely gridded structure using a personal computer.

Thus, the first abstraction necessary in modelling a building component is to approximate the geometry of its elements within a strictly right-angled system. It is advisable to prepare a simple, dimensioned sketch of the object to be analysed before beginning actual work with the program. Generally it can be said that simplifications made in the process of abstraction should be based on thermally relevant criteria in order to ensure that the model is thermally equivalent to the "real thing". Though there are no hard and fast rules on developing the ideal model (each concrete object must be treated individually), certain principles and guidelines should be adhered to. Valuable hints are settled within [standards - see also EN ISO 10211](#).

If it can be otherwise avoided, non-right angles in the elements of a building structure should not be approximated by "stepping" contours orthogonally in such a way as to severely increase their complexity. A more complicated contour generally corresponds to an increase in the surface area of the volume delineated, thereby influencing a key factor in determining heat flow patterns. Therefore, non-orthogonal geometries should be simplified by simple "straightening" whenever possible, that is, whenever the qualitative effect on heat flow can be expected to be negligible.

When stepping contours has been used to assemble the model then the increase in the surface area of the delineated volume must be adequately compensated by adjusting properties of that respective surface. Typically this is the case by increasing the value of the surface resistance R_s by the factor of the surface (or length) has grown due to such modelling decision.



Scope and scale

The scale of evaluation should be kept in mind when preparing a preliminary model sketch. This means keeping the level of detail consistent with the overall dimensions of the region to be analysed.

Though AnTherm requires dimensional data to be entered in millimetres, the total size of a particular model could also run up to tens of meters. Over detailing larger spatial structures can cause substantial increases in fine gridding, and therefore in calculation time, without the benefit of increasing the precision of relevant information provided by the evaluation branch (temperature distributions and heat flow patterns).

Finally, any thermally irrelevant elements of a building structure can and should be eliminated from the model. Such are elements which do not primarily influence heat conduction through a building component (e.g. thermally disconnected exterior elements such as ventilated siding, handrails, fixtures, etc.).

See also: [Elements of a component](#), [The component](#), [Overlapping of elements](#), [The room \(space\)](#)

Elements of the building component

In the geometric model used by AnTherm to represent a **building component**, a single rectangular region or volume in space which is defined as "filled" with one homogeneous **material** is referred to as an **element**. Building materials are defined by name (e.g. "concrete masonry unit") and thermal conductivity λ [W/mK].

For a two dimensional case it will be a rectangle parallel to coordinate system axes. The rectangle (a two dimensional interval) is specified geometrically by its extents on two coordinate axes - x_1 to x_2 and y_1 to y_2 - thus by entering four such coordinates the rectangular element is defined.

For a three dimensional case the input of further interval dimension is required - z_1 and z_2 are either provided explicitly or they result from the ordering and thickness of 3D layers.

If two such elements share the same region in space and thus they **overlap**, the element entered later "**displaces**" preceding elements (or any parts thereof).

Spaces at boundaries of a component are defined as "air-filled" elements. The air is characterised by the **surface** resistance R_s [$\text{m}^2\text{K}/\text{W}$] (or the surface heat transfer coefficient α [$\text{W}/\text{m}^2\text{K}$]) appropriate to a designated situation (e.g. "exterior", "interior/along glass", etc.). A single space must later be assigned one, unique temperature condition (in the evaluation branch).

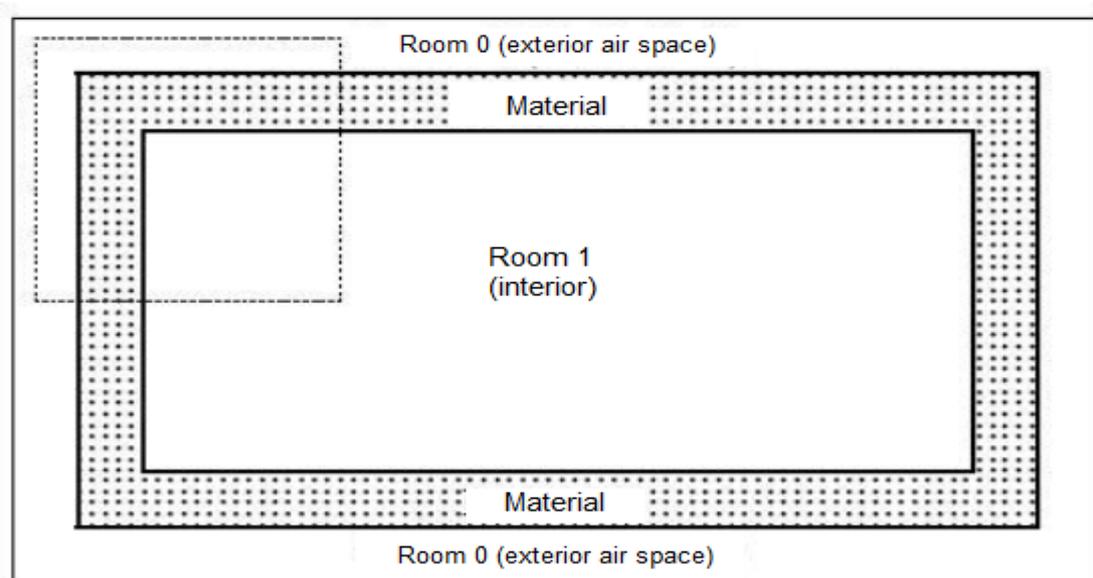
The necessary information about **material** and space **surface** properties can be found in standard literature on this topic. This information should be at hand before actually sitting down to work with the program.

Heat sources are defined by an element overlapping one or many other material elements - the name of the heat source is used to identify it (a heat source to which no power is assigned in the evaluation stage is treated just as any other non-emanating building material).

See also: [Element type \(kind/type of a element\)](#)

The definition of a component is provided by the entry of an ordered sequence of elements - these are stored into an element list providing the **order of element overlapping**.

An example shall show the definition of elements to build a simple component - walls surrounding a space. For the sake of simplicity any openings are not concerned and we assume walls to be homogenous (built of one material).



Entry of the component including all surrounding spaces is done in the most efficient way by starting the input by the entry of the exterior space (exterior air) rectangle. Any arbitrary size of the rectangle shall be chosen - it is sufficient if it extends beyond the exterior of walls which itself surround the interior space.

Following the entry of the exterior space the wall material rectangle is entered at its exact (exterior) dimensions. Finally, the third element, the interior space is entered - this results in **overwriting** the partial region of the earlier entered material element. The entry is finished.

Of course one could enter the model in some different (circuitous) way: all three different part types - exterior air, material and the interior air - could be entered as non overlapping rectangles. This would require much higher number of elements to be entered however.

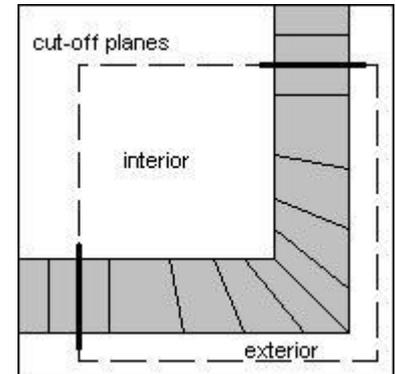
Size reduction and adiabatic Cut-Off planes

Although entire buildings can also be evaluated by AnTherm, most practical applications will not be so global in nature. Usually only spatially limited problem zones need be scrutinised. Such a problem zone is typically a structural detail or region which could prove thermally critical.

Since thermal bridges are characterised by two- and three-dimensional heat flow patterns, regions of a building structure in which heat is expected to flow one-dimensionally (i.e. perpendicular to material surfaces) are generally of little interest in using AnTherm.

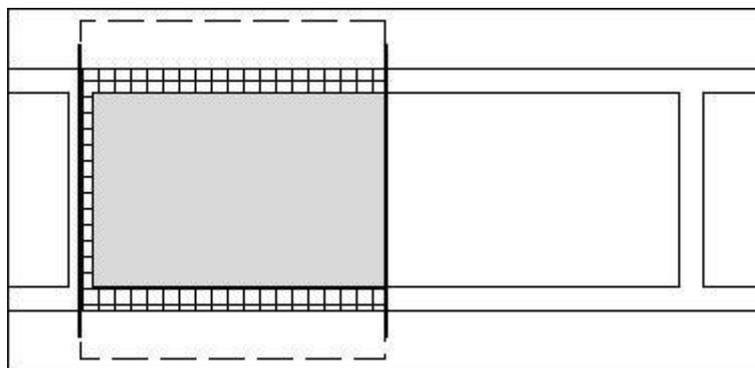
Therefore such regions may be eliminated from the model to be considered without affecting any relevant results. The outer boundary of the region to be analysed is defined by cut-off planes. These planes are idealised as a non-conducting surface and should therefore be placed parallel to the direction of heat flow in a region where one-dimensional heat flow can be assumed.

Just where to optimally draw the cut-off planes - the nearest regions of a problem zone where heat can be expected to flow reasonably normal to planar component surfaces - is initially an educated guess, which may, of course, be subject to revision upon evaluation.



To reference the earlier example the input shall be accomplished in the same manner - at first the exterior space is entered followed by the material rectangle later excavated again with the interior space rectangle. The most important difference to the example above is in that the boundary of the component is only partly covered by space surfaces. The other parts of the boundary (emphasized in bold) are cut-off planes implicitly resulting from the arbitrary user choice to cut the thermal heat bridge at such boundaries, thus inhibiting the heat flow completely (the normal component of heat flux through a cut-off plane is nullified). In this example placing cut-off planes far enough from the corner will be sufficient.

One can also draw upon qualitative characteristics inherent in the geometry of a given object to limit the region to be modelled. Particularly axes of local symmetry in a building component, which can be safely assumed to coincide with planes parallel to heat flow, should be used to cut down the size of a model wherever possible.



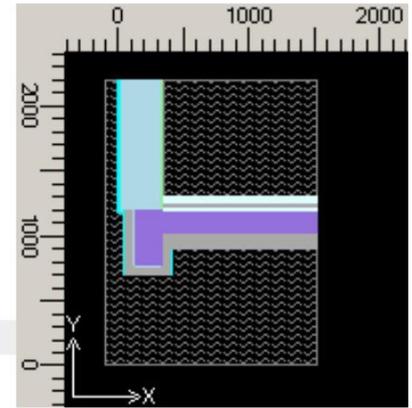
If the model shown in the image above is thought to continue to the left also two types of symmetries can be identified. These are not only the geometrical symmetries, but also the symmetry of temperature distribution which can be expected easily. Due to such symmetry there will be no heat flow through such line. The analysis of such model could be reduced to the area between the two symmetry lines.

See also: [The Component](#)

The Building Component

The model of a building component is, generally put, built up with rectangular, possibly [overlapping, elements](#). The term "component" need not have to be associated with typical architectural or engineering meaning of building assemblies. The "component" might resemble an excerpt from the building assembly containing one or more building components in its typical meaning - like ceiling junction, window frame connection etc.

Within that application the term "component" does simply mean the [ordered](#) sequence of input [elements](#) composing some arbitrary object. The convenient input principle inherent to AnTherm - [element overlapping](#) - can facilitate construction input considerably if employed properly.



Groups

The construction to be simulated might contain some repeatedly appearing details.

The input branch provides the notion of such details by grouping them together (by unique group names) providing the method of quick selection, translation, rotation (by multiple of 90°) and mirroring (at axes parallel planes).

The ability to compose any construction from groups is provided as an additional helpful method significantly simplifying the input process.

Three dimensional constructions

The input of three dimensional components follows same rules as those for two dimensions.

3D Layered Model

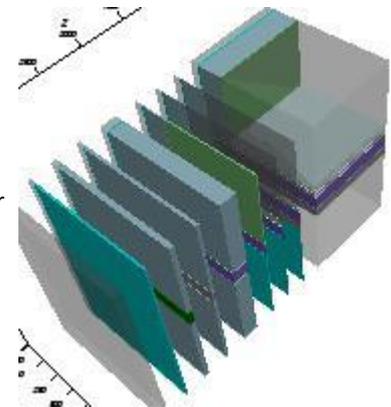
Three-dimensional objects are modelled as a "sandwich" of parallel layers. Each layer is composed of one or more elements. The elements of layers are entered to form a two-dimensional description (x and y axes), or section, normal to the direction chosen for layer depth (z axis).

The [thickness of a particular layer](#) is defined by the depth for which a given section is a valid description of the structure. In other words, a separate layer must be specified for every change in cross-section when working through the layers of a particular model.

This should be taken into consideration from the very beginning in the form of orientation sketches showing the number of layers (x/y sections) which will be needed in the z direction.

Objects for which a two-dimensional analysis will suffice are modelled as an x-y section and evaluated per meter in the z direction. A dimensioned orientation sketch should include the origin which will be used for x and y coordinate input (an origin position along a component surface is usually the clearest solution).

See also: [Project types: Layered 3D Project](#)



3D Model

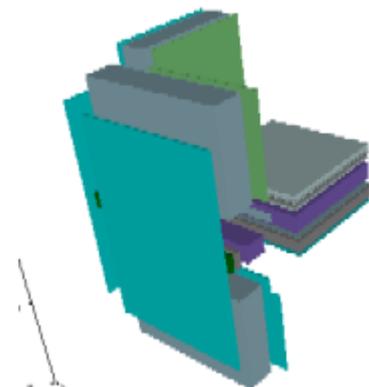
The 3D Model is created by entering arbitrary triples of coordinates.

The component's geometry is therefore described by input of ashlar.

The resulting final 3D project is a sequence of such ashlar.

The order of ashlar is significant in that ashlar (i.e. three dimensional elements) sharing same regions overlap, and those entered later hollow the properties of the others - by the same [overlapping principle](#), as with two dimensional rectangles.

See also: [Elements of a component](#), [Overlapping of elements](#), [Project types](#)

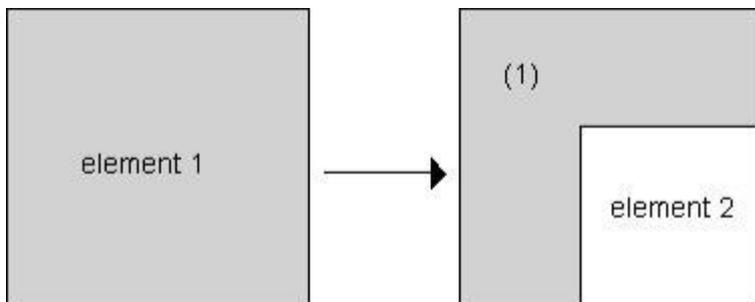
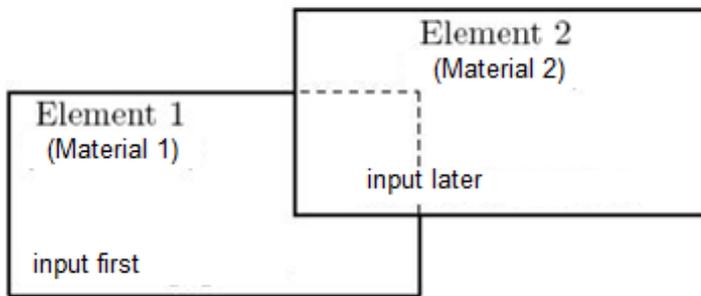
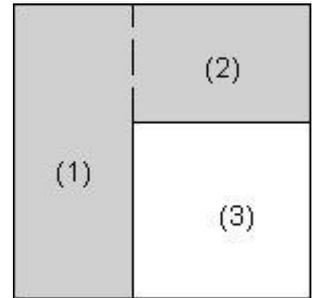


Element overlapping

Elements of a shape more complex than a simple rectangle can be built up of sub-elements of the same material according to an additive principle. However, in many situations, a simpler subtractive principle inherent to AnTherm can be implemented to achieve the same geometry more easily: *element overlapping*.

The principle of element overlapping means that any element entered "displaces" preceding elements (or any parts thereof) which share the same region in space.

Thus, for example, a shape such as the corner situation illustrated here can be drawn either as a composite of three elements or with a series of two over-lapping elements:



Which element over-rides the other depends solely on the sequence of element input, as reflected in the [element list](#).

Warning! Changes to the sequence of elements in the [elements list](#) will result in different effect of overlapping.

See also: [Elements of a component](#), [Element Browser window](#)

The Space (Room) and the Surface

While defining the modelled building component the entry of spaces (rooms) are the core part of the input, especially the input thermal transfer resistance R_{si} and R_{se} [m^2K/W] (or thermal transfer coefficients α_{si} und α_{se} [W/m^2K]) of their surfaces appropriate to a designated situation (e.g. "exterior", "interior/along glass", etc.).

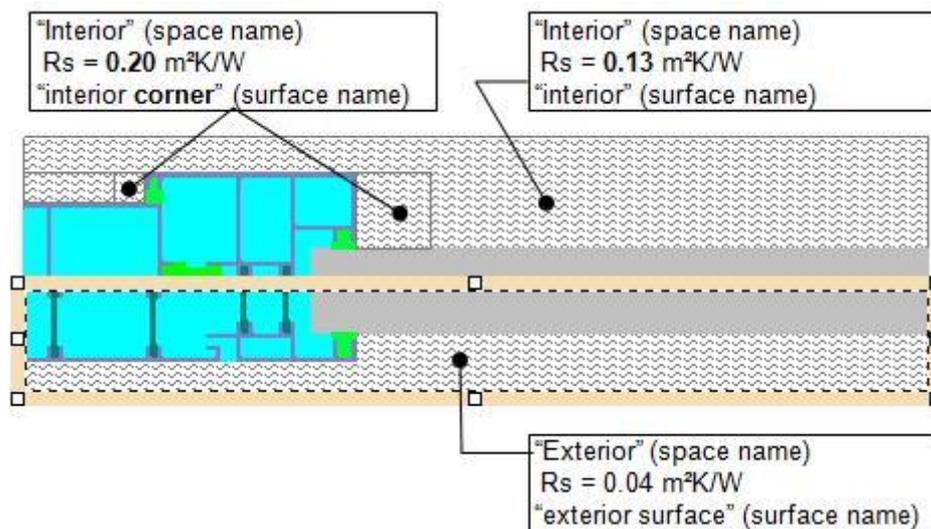
Spaces at boundaries of a component are defined as "air-filled" elements of equal temperature (and relative humidity); in the sense described here, exterior air is to be treated as a "space" too.

Distinctive spaces are identified by their unique names ([Space Name](#), e.g. "exterior", "interior", etc.). A single space must later (not in input, but in the evaluation branch) be assigned one, unique [temperature condition](#) (and eventually relative humidity).

The "space" as described here need not be locally coherent but it shall be composed of intentionally separated areas; If all such areas receive identical [space name](#) they will be treated as one, coherent, space.

Surfaces of different spaces shall not be connected directly to each other, i.e. without placing some material construction interconnecting them.

The space may, however, be divided into several elements to account for different [surface](#) situations within the space. Such "sub-spaces" can then be identified with different transfer resistance (or transfer coefficient) values ([surface property](#)) but sharing common space name.



See also: [Elements of the building component](#), [Element overlapping](#), [The heat source \(power source\)](#), [Element type \(kind/type of a element\)](#)

The heat source (power source)

The "heat source" describes an spatial area of a component heated with constant power. Distinctive heat sources are identified by their unique names ([Power Source Name](#), e.g. "heating stripe"). The [specification of the heating power](#) of the source is postponed until the evaluation of results is executed.

One heat source in the sense described here can be composed of several, spatially not connected ranges; if all such ranges are assigned same name (heat source name) than all these areas are considered one source.

Different heat sources shall have common boundaries if required.

See also: [Creating the grid](#), [Elements of a component](#), [The Space \(Room\) and Surface](#), [Element type \(kind/type of a element\)](#)

Creating the Grid (Gridding)

Having the input branch completed and prior to obtaining the *results* crucial decisions have to be taken having major impact onto the precision of results and calculation time required to create them. The principle behind the calculation method of AnTherm lies in balancing the thermal characterisations of orthogonally subdivided regions. Each subdivision, or *cell*, is thus geometrically bound by orthogonal *grid planes*; these must be defined such that each resulting cell contains only a single building material. Thus, the resulting subdivisions need not to be identical with the input *elements*. The process of creating proper subdivisions is called "*gridding*".

Minimal Grid

The coarsest grid structure which satisfies the requirement of homogeneous cells is referred to as the *minimal grid*. This can be derived directly from the *geometric data (elements)* of a *model*. This minimal grid is generated fully automatically without any further user intervention.

Fine Grid

Since a minimal grid is generally too coarse a subdivision to provide dependably accurate results by numerical methods, a further - finer - gridding of the model must be performed for calculation. AnTherm offers automatic method for generating such auxiliary grid planes. The minimal grid thus generated forms the basis for all further gridding, but it does not constitute a complete grid structure. A *fine grid* in some form must be generated and stored in a grid file before moving on in the calculation branch.



The creation of a fine grid involves inserting additional grid planes. Such procedure might be very circuitous and tedious when done manually. Therefore AnTherm provides an automatic generation of the fine grid controlled with few *parameters*. The automated process ensures, that most critical regions (corners, edges, etc) will receive finest grid.

The fine-gridding defines the final set of cells (orthogonally subdivided regions) which shall then be balanced. The number of such cell might be very large! Typical 2D calculation will result in 20.000-50.000 cells to solve with sufficient precision. 3D calculation will quickly grow to 1.000.000 cells and more!

The gridding (discretization) algorithm of AnTherm subdivides the component model under consideration into cells which will be thermally balanced thereafter. The calculation algorithm creates for each such cell the heat balance equation and then solves the system of such equations. The solution results describe temperature field with values in the middles of cells.

In the event that a minimal grid would suffice for calculation (e.g. for preliminary evaluation of a model), it can be re-generated as a fine grid based on the appropriate parameters.

Super Fine Grid (The "super" - refinement)

Considering places where temperatures will later be calculated (during the evaluation) not only middle points of each cell are of interest - middles of cell sides and cell edges and corners of each cell will be calculated too.

Such *super-fine-gridding* provides for the two dimensional calculation of 20.000 cell about 80.000 evaluated temperature points. Within a three dimensional case with a fine grid of 20.000 cells the evaluation will result in more than 160.000 temperature values. If the fine grid size is large - e.g. 1.000.000 cells for a 3D case - the number of calculated temperature points will grow to more than 8.000.000 possibly requiring longer calculation times during the evaluation. For that reason AnTherm will display a warning message when the number of cells within the fine-grid grows to higher value. In such case it is worth to consider, if, by adjusting some fine-grid parameters resulting in more coarse grid, sufficient precision can be reached faster also.

As the nice result of such refinement the provisioning of much higher number of node values is available for the evaluation, hence the evaluation can be much more precise thereafter - for the two dimensional case the number of temperature nodes is fourfold the original cell count and in three dimensional case it is eightfold! For example, the calculation with 1.000.000

(one million) balanced cells provides results of thermal balance equations

at more than 8.000.000 (eight million) points! - and the operation is accomplished in surprisingly short computation time. In the event that a minimal grid would suffice for calculation (e.g. for preliminary evaluation of a model), it can be re-generated as a fine grid based on the appropriate parameters.

The Calculation (Solver)

Besides the knowledge of geometrical and physical composition of a building construction the calculation the temperature distribution within a building construction requires the knowledge of [particular boundary conditions](#) also. The approach used in this program is founded on top of specific boundary conditions - those defining the air temperature of each space connected to the construction. At cut-off planes an implicit adiabatic boundary condition is applied automatically by forcing the normal component of heat flux to zero, thus no additional user input is required. Therefore the user needs to provide particular air temperatures only. It is to be considered that only uniform air temperature shall be assigned to particular space, thus effects like variation of temperature due to air flow layers cannot be input directly. If heat sources are present within the construction additional boundary conditions defining the heat density of each particular source have to be supplied also.

A characteristic of the program is the division of the procedure for generating a temperature distribution into two stages: calculation of base solutions and evaluation under prescribed conditions.

Heat flow in a building construction, as well as the subsequent temperature distribution, can be described mathematically by differential equations. Most importantly, these equations are linear and homogenous by nature. This means that one set of solutions, calculated for a specific set of conditions, can be "re-used" as the basis for solutions under differing conditions by superposition, i.e. by linear combination of selected base solutions (see "[Theoretical background](#)"). More specifically, base solutions are calculated under the assumption of a "basic" set of boundary conditions: with an air temperature of 1 in the selected space, and 0 in all others.

The calculation approach in the program uses this circumstance to minimise over-all evaluation time for user's comfort. One set of base solutions need be calculated only once to characterise a given model, which can subsequently be considered under varying conditions without repeating the time-consuming computation necessary to solve the primary set of differential equations. The superposition under applied boundary conditions is postponed to the evaluation part, thus the determination of basic solutions in **that step does not require any input of particular boundary conditions**.

Computation time is further reduced by utilising the weighting function character of base solutions (normalised such that their sum must equal 1). Hence, if N cases have been selected, only N-1 solutions need to actually be calculated. The N-th base solution is then very simply derived as a difference of the sum to 1, that is, by a separate stage of superposition.

If heat sources are present within the construction too, respective base solution will be created for each particular heat source also. The calculation is provisioned by setting the heat density of particular source to 1 while leaving all other boundary conditions at 0. Therefore the total number of base solutions calculated is the sum of the number of spaces and the number of heat sources.

The [parameters of iteration](#) can be manipulated by the user prior to a calculation run of course. This is of interest when a model is particularly complex or when base solutions which have already been calculated prove to be inadequately precise during subsequent evaluation.

See also: [The Over-Relaxation](#), [Theoretical background](#), [Results & Evaluations](#)

The Over-Relaxation

In general there is no explicit need for additional input during the calculation. The user can control the solver by [choosing various parameters of the computation](#). Such most important control elements shall be described in short below.

Generating a temperature distribution is connected with the numerical solution of a very large linear system of equations. Such an equation system is solved through the relaxation method [\[2\]](#). The relaxation factor ω must be between 1 and 2.

Relaxation factor ω

The value of the [optimal relaxation factor \$\omega_{opt}\$](#) , which leads to the most expedient convergence of the calculation method, is highly dependant on the size and conditioning of the equation system to be solved. This value differs from case to case and is initially unknown. During the initial step of calculation an analysis of the equations system is executed and an estimation of ω_0 for the optimal relaxation factor is calculated iteratively.

Due to the fact, that the [optimal relaxation factor \$\omega_{opt}\$](#) , which ensures the fastest method convergence, can only be roughly estimated, the value calculated serves as an initial parameter for the relaxation factor ω which is then modified with each iteration in the course of the second stage of calculation. In particular, the relaxation factor grows from ω_{min} towards (asymptotically) the value of ω_{max} . If certain conditions are met it is reset to ω_{min} and the process continues.

General parameters responsible for the automated variation of ω are set to default values. Under normal condition (moderate number of cells calculated, absence of extremely high or low conductance values) the calculation will appropriately execute without any user intervention.

If the convergence might be slow in exceptional case the user can speed up the convergence by changing the default values. The major indication is provided if the number of cell is very high and the optimal value of ω is just below 2. Detailed description of [\$\omega\$ -control parameters](#) can be found in the separate section of this manual.

Solving the system of equations

At each iteration step the method determines the deviation of the value calculated for each network element (cell) compared to the previous iteration. The maximum of all absolute values of these differences is used to control the convergence and progress of the calculation. If that value continually is below some defined threshold ϵ during several successive iteration steps, the [termination condition](#) is considered fulfilled and the calculation finished.

During the [evaluation](#) the user shall check if all precision criteria are satisfied and, if required, he can reduce the termination condition parameter ϵ and enforce the calculation to continue.

The computation can be interrupted at any time (e.g. if the computing device is needed for some other purpose) to be resumed later. The calculation will resume starting from some intermediate results saved at the time of interruption.

Complete results of the calculation are written to [several files](#), which are then required during the evaluation. There are no special output results shown available for printout during the calculation stage.

See also: [Theoretical background](#), [Solver-Parameter](#), [Application settings](#)

Evaluation of results

Final output data is obtained in the evaluation branch, during which the model under consideration (or rather, its characterisation through base solutions obtained from the calculation) is evaluated under specific thermal boundary conditions.

While thermal coupling coefficients, heat source distribution and weighting factors are boundary condition independent, further output data is dependant on air temperatures of spaces calculated and, in the event that heat source elements have been included in the model, power density values, which all must be assigned as a boundary conditions to these respective base solution cases as well.

Thermal coupling coefficients matrix and heat source distribution factors

Thermal Coupling Coefficients [W / K]

Space\Space	aussen	Hallenbad	Wohnraum
aussen		1,386223	1,292194
Hallenbad	1,386223		1,476608
Wohnraum	1,292194	1,476608	

Thermal Coupling Coefficients (L2D) [W / m²*K]

Space\Space	aussen	Hallenbad	Wohnraum
aussen		0,373884	0,518612
Hallenbad	0,373884		1,211041
Wohnraum	0,518612	1,211041	

As the direct result of the computation the matrix of [thermal coupling coefficients](#) is provided. For a two dimensional model the matrix shows length related transmittance L^{2D} [$Wm^{-1}K^{-1}$]; for three dimensional construction the matrix displays thermal coupling coefficients L^{3D} per se [WK^{-1}]. Because thermal coupling coefficients – and as such elements of the conductance matrix - are independent from particular boundary conditions, the matrix itself can be output prior to any user input of temperatures.

Based on the output of thermal coupling coefficients one can calculate the thermal transmittance values (linear thermal transmittance Ψ (Psi) for two dimensional linear thermal bridges (see [Psi-Value Determination](#)), or point thermal transmittance χ (chi) for the three dimensional point thermal bridges).

In the event heat sources are available also, the distribution factor of each source is shown too - like the thermal coupling coefficients this values are independent from boundary conditions also. If N spaces are attached to the considered construction the distribution table will shown N numbers. The i -th ($i = 1, N$) value of the distribution table shows the percentage of the heat provided by the particular heat source passing to the i -th space. The values of the distribution table are therefore from the range 0 to 1; because the steady state calculation does not cover the heat capacity storage, the sum of all distribution values must result in 1 (apart from minor rounding errors).

Dynamic, transient problems, in which the heat storage capacity is taken into account, result in the output of [harmonic, periodic coupling coefficients](#), also calculated directly and not dependant from any particular boundary conditions, are output as a matrix for each specific period length chosen. This results can be used for the analysis of dynamic behaviour of the component for example to read out amplitudes and phase shifts / time lags (like L_{pe} needed within Passivhaus Projektierungspaket PHPP) or to calculate effective mass capacities.

Boundary conditions

Any further evaluation requires the definition of [boundary conditions](#)- these are specified by air temperatures of spaces connected to the building component and all power densities of all heat sources. Only after that data has been provided further results can be requested.

Boundary conditions and Temperatures / Humidity

	Air temp. [°C]	min.temp. [°C]	max.temp. [°C]	Condens. RH [%]	f_{rel}
Room 0	-10,00	-9,97	-9,43	100,00 %	
Room 1	20,00	15,22	18,49	73,96 %	0,84

Power source	Power [W]	Power density [W/m ²]	Volume [m ³]
PS 0	0,00	0,00	0,01176000

Temperature extremes, dew points, f_{RSi} -temperature factors, g -values

As soon as the [boundary conditions](#) have been applied, the application determines locations on the surfaces of all spaces of the model at which the minimum temperature is reached for given [boundary conditions](#). The report "[Results](#)" lists coordinates of such points together with the value of the temperature at that point, the dew point value [\[1\]](#) resulting from that temperature and the corresponding temperature factor f_{RSi} . The temperature weighting factors (" g -values") are also output for these coldest points. It is worth to mention, that the temperature weighting factors are independent from the boundary conditions. Contrary, the location of coldest surface points is dependant on the boundary conditions chosen, and therefore the output of weighting factors for coldest points can be provided after boundary conditions have been set.

Weighting factors for coldest surface point

	Room 0	Room 1
$g(PS\ 0)$	0,000129	0,065448
$g(Room\ 0)$	0,999128	0,159403
$g(Room\ 1)$	0,000872	0,840597

According to the standard conformance requirement of EN ISO 10211 the "[Results report](#)" can be output to the printer.

For some special purpose it might be desirable to receive some output of the distribution of g -values within the building construction or at its surfaces. This is possible within the evaluation stage also. By using the nature of g -values, which in particular are base solution values, the user has to assign the boundary condition of the base solution of interest to 1 to receive the kind of "temperature distribution" showing the g -values themselves. The output can then show isolines, false coloured images or visualizations of "surface g -values" for example.

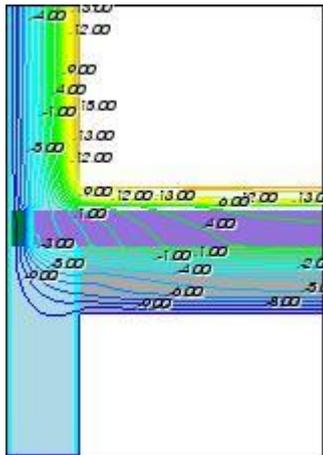
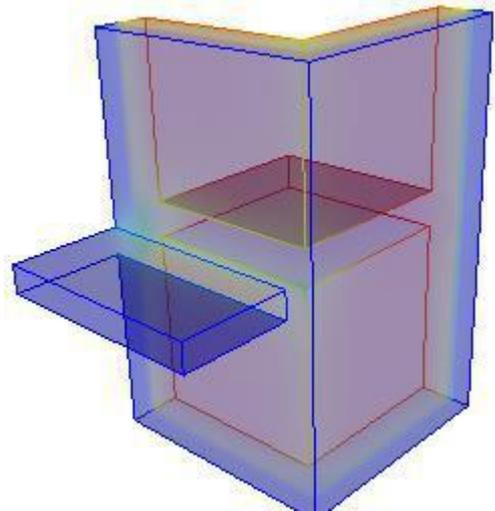
Finally, the output of f -values is also possible because those are special case of g -values.

See also: [Evaluations and Results](#)

Visualization - Graphical Evaluation

Along with the [numerical output](#) formally required by EN ISO 10211 further extensive [graphical evaluation \(visualization\)](#) of the calculated results can be obtained and eventually passed to printer or other applications.

Quick view onto the calculated temperature distribution of the component can be obtained in the form of "false coloured" visualizations. For a two dimensional case there is only one such image, three dimensional cases can be visualized with slices through the model or arbitrary view onto the construction.



Some more detailed representation of the temperature distribution is available as [isotherm images](#). Isotherms, i.e. lines of onstant temperature, will be shown on the slice through the construction for a two dimensional model. Detailed appearance and content of the information shown can be defined by the user by defining the number of isotherms drawn - value of the first and last isotherm, interval between consecutive isolines etc. can be used to define the meaning of each particular isotherm..

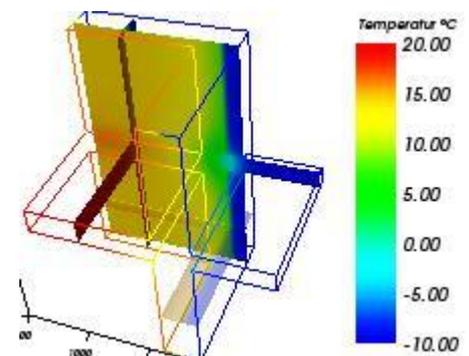
For a three dimensional case the [slice plane](#) at which isotherms shall be drawn can be arbitrarily chosen. Furthermore isotherms can be shown on [the surface of the model](#) to emphasize the temperature distribution also.

Furthermore, one can query the value of temperature at any arbitrary point of the construction. The definition of points of interest is either provided by the coordinate entry via the menu "[Probe points](#)" or by setting the position of [slice planes](#) - the [temperature value at the intersection the three planes](#) will be shown on the screen.

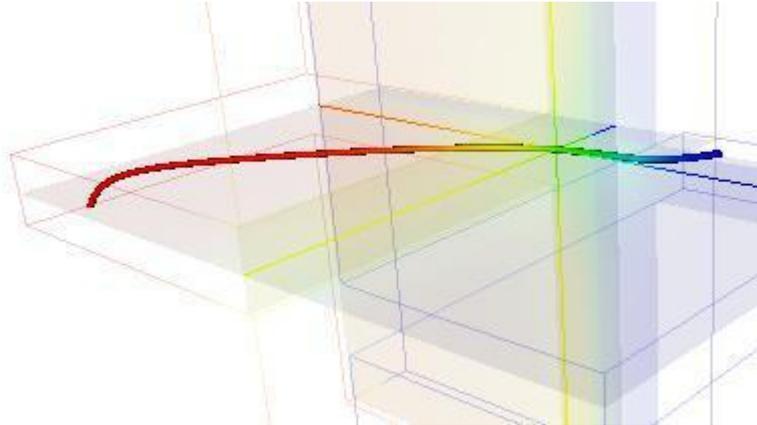
Further evaluation of temperatures can be available as a graph of temperature values along an edge (surface) of one particular space in two dimensional case. (*Remark! This option is currently not available!*)

For three dimensional cases one shall request the [graph of temperature values](#) along arbitrary line (eventually parallel to coordinate axes). Such line might intersect the interior of the component or follow the surface of the construction - thus providing the ability to emphasize temperature values at components edges.

As an alternative to the visualization of calculated temperature distributions one shall request results showing dew point values (condensing humidity values) resulting from the current temperatures on the surfaces of the model. To achieve that simple activation of the switch "[secondary functions](#)" is required followed by the selection of "Relative Humidity %" within the box "[Active function](#)". This visualization provides by far much more information compared to the norm required printout "[Results](#)" because not only the "yes-no-decision" of norm conformant construction can be received, but additional, much more detailed conclusions about the distribution of dew point values at component's surfaces are possible. In particular, for construction non conformant to standards, the ability to draw an [isoline](#) at the humidity value which is critical by the standard (the value above which the condensation or mould growth shall occur) might be very instructive. Such an [isoline](#) will mark and emphasize the critical area of the components [surface](#).



Very helpful hints regarding improvement of the thermal quality of building construction result from the visualization of heat streams through (and within) the construction. To receive output related to heat flux one simply needs to activate the switch "[Secondary functions](#)" and then, within the box "[active function](#)" choose the "Heat stream density W/m^2 ". Variants of visualization options available to the calculated heat stream densities are the same as to temperature distribution. Locations of high heat stream densities shall emphasize thermally weak areas of the construction and can be immediately visualized as false colour diagrams or [isoline views](#) for better identification.



For both, the two as well as for three dimensional cases, the heat flow through the construction can be visualized with a streamline. The streamline is shown as soon as the control panel "[Streamlines](#)" is activated and coordinates of the starting point are chosen by placing the intersection of [the slice planes X, Y and Z](#) within the construction.

For two dimensional cases there is possibility to visualize multiple [multiple heat flow streamlines](#). Since the area between the two neighbouring streamlines marks exactly the same amount of heat flow, the areas at which the density of streamlines is high emphasize potentially weak locations within the construction.

Visualisation of heat flow is also offered as [hedge hog field of vector arrows](#). This display is controlled within [Vectors \(HedgeHog, Arrows\) control panel](#) by defining the mesh of points at which the direction of heat flow and eventually its magnitude shall be shown.

See also: [Evaluations and Results](#)

Multidimensional Vapour Diffusion

Computational description of vapour diffusion in building constructions can provide valuable hints for answering the questions if there is a risk of, potentially destructive, interstitial vapour condensation within the core of the construction or not (Interstitial Condensation Assessment).

Diffusion equations have the same structure as those describing heat transfer and can be solved analytically only for one dimensional vapour diffusion. In all higher dimensional cases numerical methods, like the one implemented in AnTherm, have to be used.

VAPOUR-option: Analysis of multidimensional vapour diffusion is only possible with an active VAPOUR-Option of the program.



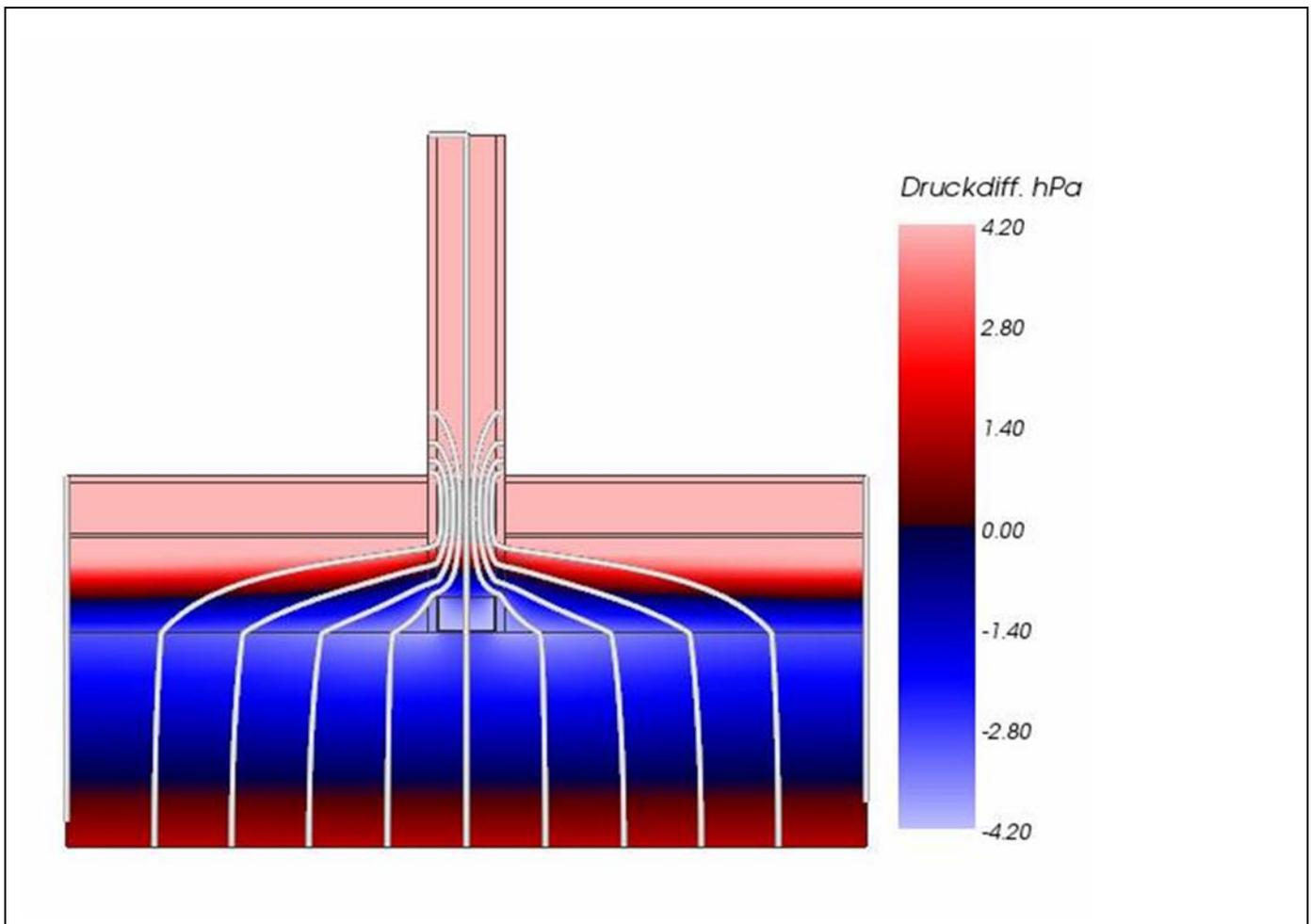
AnTherm calculates two- and three-dimensional distribution of vapour partial pressures in building constructions.

Saturation pressure distribution within the construction is computed too - based on the temperature distribution.

The comparison of the partial and saturation pressure distributions provides hint directly related to answering the questions if there is quantitative risk of condensation within the construction or not.

All areas at which partial pressure of vapour is higher then the saturation pressure can be accounted as endangered and can be visualized as such graphically (e.g..shown as **false colours**, or as **isosurface** or an **isoline** at the zero value of the **difference of vapour pressures** or one specific value of **relative interstitial core humidity**).

Example: Vapour pressure difference and vapour diffusion streams



This humidity model calculation shows, due to the cold bridge effect caused by a partition wall of stud framing gypsum plaster placed directly onto the slab on the one hand, and - due to the interruption of the moisture barrier on the war side, just below the heating pavement - the possibility of diffusing water vapour into the floor structure through the partition wall on the other, additional regions of higher condensation risk. These are highly distinctive in the surrounding of the bottom U-Profile of the partition wall.

Legend:

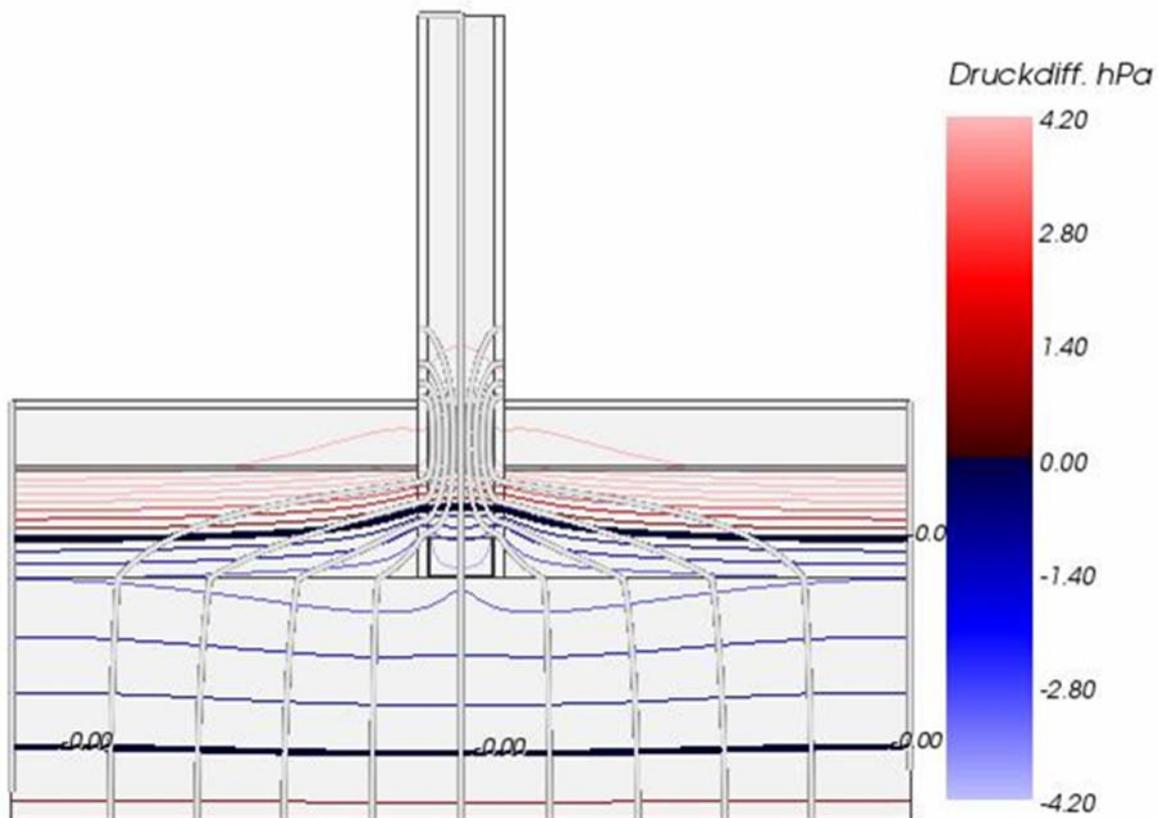
- colour mapped vapour pressure difference (black:zero-boundary; blue:negative; light-blue/white:highly-negative)
- stream lines of the vapour diffusion (white) at 10%-interval of the overall vapour diffusion stream from inside (top) to outside (bottom)

Transposing this method used in AnTherm onto the commonly used Glaser-Method for one dimensional case means, that intersections of vapour partial pressure distribution with saturation pressure distribution are calculated and regions within these intersections are interpreted as condensation risk areas. It is well known that the width of such zones is mostly overestimated.

It has to be accounted, that when interpreting such results, diffusion is not the only driving factor for vapour transport. Especially within massive constructions water and vapour transport can be significantly influenced by capillary transport hence diffusion might be less significant then.

The implementation provided in AnTherm shall be mainly used for answering **the question if, and if so where within the construction, by given boundary conditions (air temperature and relative air humidity) moisture is produced.**

Example: Vapour pressure difference and vapour diffusion streams



This humidity model calculation shows, due to the cold bridge effect caused by a partition wall of stud framing gypsum plaster placed directly onto the slab on the one hand, and - due to the interruption of the moisture barrier on the war side, just below the heating pavement - the possibility of diffusing water vapour into the floor structure through the partition wall on the other, additional regions of higher condensation risk. These are highly distinctive in the surrounding of the bottom U-Profile of the partition wall.

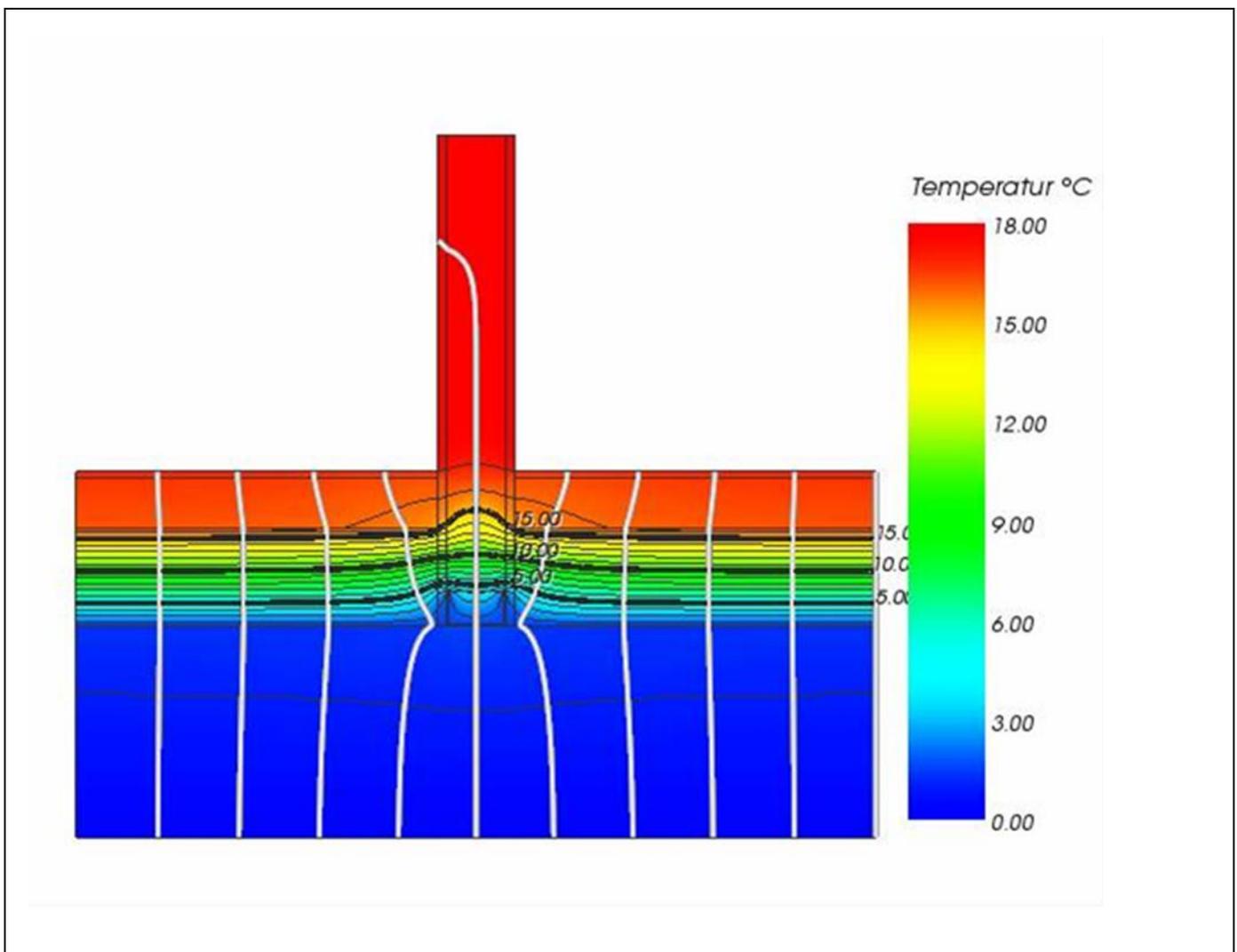
Legend:

- Isolines of vapour pressure difference (black);
- stream lines of the vapour diffusion (white) at 10%-interval of the overall vapour diffusion stream from inside (top) to outside (bottom)

Remark: More exact estimation of condensation zones and moisture amounts is subject for future developments.

See also: [Theoretical background](#), [European standards on thermal bridges](#), **EN ISO 13788**

Example: Distribution of the temperature and heat stream lines



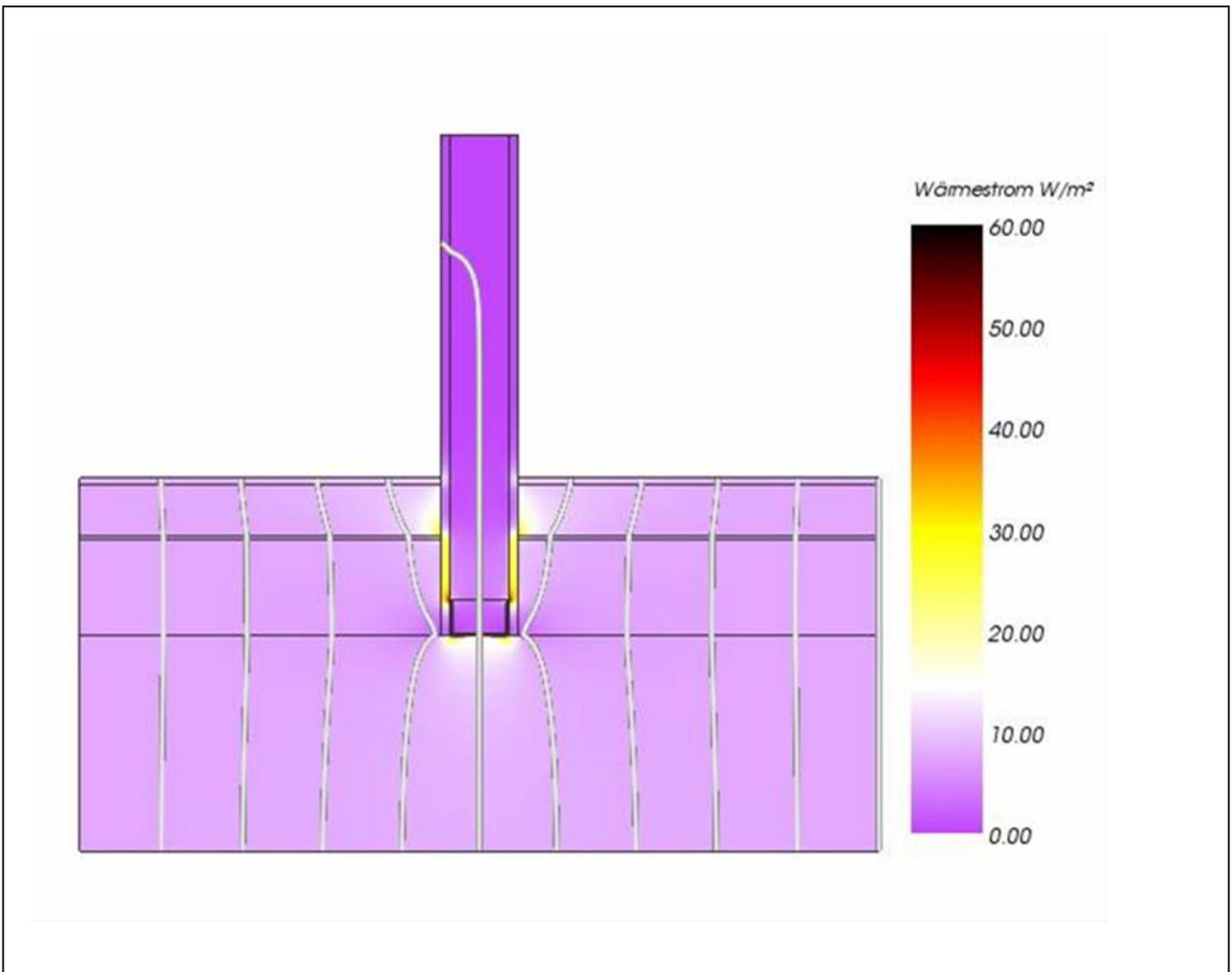
Partition wall of stud framing gypsum plaster placed directly onto the 30cm thick concrete ceiling above basement garage, without thermal insulation on the cold side.

The EPS heat insulation layer (Röfix 831, zementgebundene Polystyrolflocken) of 13,5 cm thickness has been placed solely on top of the concrete plaster - there is no other heat coating at the bottom side of the ceiling. The typical cross section of this ceiling construction has been assessed being not critical at all.

Legend:

- isotherms (black);
- colour mapping by temperature
- heat flow streamlines (white) at 10%-interval of the overall heat stream from inside (top) to outside (bottom)

Example: Heat flux distribution and heat stream lines



Partition wall of stud framing gypsum plaster placed directly onto the 30cm thick concrete ceiling above basement garage, without thermal insulation on the cold side.

The EPS heat insulation layer (Röfix 831, zementgebundene Polystyrolflocken) of 13,5 cm thickness has been placed solely on top of the concrete plaster - there is no other heat coating at the bottom side of the ceiling. The typical cross section of this ceiling construction has been assessed being not critical at all.

Legend:

- colour mapped heat flux (flow stream density) (yellow to black: heavy heat bridge impact)
- heat flow streamlines (white) at 10%-interval of the overall heat stream from inside (top) to outside (bottom)

Harmonic (periodic) dynamic indicators

The **HARMONIC** option of AnTherm calculates “harmonic thermal conductivities” of component or building as indicators required for transient building energy simulations based on periodic (harmonic) approach. This also includes PHPP calculations. Thermal Harmonic Conductivities and further dynamic indicators facilitating heat transport with heat capacity capabilities follow the ISO and EN standard EN ISO 13786.

HARMONIC-option: *Solving dynamic, periodic problems for determination of Phase Lag in response to heat capacity potential is only possible with an active HARMONIC-Option of the program.*



The harmonic thermal conductivity is given by the amplitude and phase lag of the heat flow which follows periodically changing boundary conditions (either temperatures or heat sources or sinks). Lagging in time with damping of amplitude (attenuation) results from heat storage capability of materials.

Heat losses might lag many months, intermittent heat gains might result also. Cooling infrastructure might become obsolete or overheating can be practically resolved with the building own (heat) capacity.

Transient heat flows are heavily dependent on heat capacity of building components, especially massive, solid or brick walls, slab or ground structures.

The Harmonic option of AnTherm can be seen as a small brother of the much more extensive [TRANSIENT-Option](#). The purpose of the Harmonic option is to solve transient, periodic heat transmission equation for the 0-th and 1-st harmonic. The 0-th harmonic is the steady state solution of the equation corresponding to mean heat flows under periodic boundary conditions. The 1-st harmonic provides solutions of amplitude and phase lag for the main period, such as day or year.

When power sources are involved AnTherm’s Harmonic option reports Harmonic Heat Distribution Factors (additionally to Harmonic Thermal Conductivities reported for spaces) and are also given by their amplitude and phase lag of the heat flow following the oscillation of respective heat source or sink.

By using the harmonic indicators much better approximation of dynamic thermal behaviour of the building can be obtained when compared to simplified steady state approach – even with only one first harmonic of the oscillation.

Exact indicators calculated with AnTherm for any 3 dimensional geometry and arbitrary combination of materials are indispensable replacement for approximations restricted to few simplified geometries only as those provided in ISO 13370.

The strength of the HARMONIC tool relies on the fact, that harmonic thermal conductances (complex number, amplitude and the phase lag) can be calculated without the need to enter nor to know any boundary conditions – harmonic thermal conductances are output just following the steady state thermal conductances L^{2D} , L^{3D} in the respective report. There is also no need to specify or assume any starting boundary conditions as compared to time step based methods.

It is very important to emphasize, that harmonic thermal conductance are characteristic for the object concerned and by that are independent of any boundary conditions. The proper calculation of these harmonic periodic indicators does not need to know nor to apply any boundary conditions. Further evaluation of the behaviour of a building component under certain and periodically changing boundary conditions is subject to the “[TRANSIENT](#)” Option of AnTherm (higher harmonics, periodic boundary conditions, animation in time and timeline outputs).

Remark: The HARMONIC Option alone basically offers evaluation under steady state boundary conditions (in addition to harmonic, periodic, dynamic indicators).

Remark: The numerical results (stored within [the folder of simulation results](#)) shall be passed to the Program THESIM (www.thesim.at).

See also: [Theoretical background](#), [European standards on thermal bridges](#), EN ISO 13786, EN ISO 13370, [Transient](#), [multidimensional calculations by an example of components in contact with soil](#)

Transient (periodic) time dependant simulation

The **TRANSIENT** option of AnTherm calculates, evaluates and visualizes heat flows and temperature distributions in building components under boundary conditions changing in time periodically. By that it allows modelling and analysis of effects of heat storage in building construction and their response to changing conditions due to heat capacity of materials.

Such simulation approach is also known by “Transient periodic”, “Transient harmonic” or “quasi steady periodic”.

The efficiency and precision of this simulation approach results from the assumption of periodicity. By assuming periodicity of the processes the proper of periodic simulation does not need any starting distribution of temperature field. This is especially true for climatic boundary conditions (year, day), utilization scenarios (including the period of one week) or any other “repetitive” situation, e.g. periodically turning on and off electric heating appliance.

TRANSIENT-option: *Solving and evaluating time dependant dynamic, periodic problems when heat capacity effects are concerned is only possible with an active TRANSIENT-Option of the program.*



Respective boundary conditions, periodically variable in time, can be modelled as series of values at specific time points or as series of mean values valid for several intervals of equal or variable lengths. Modelling periodic boundary conditions is not limited to air temperature but also possible for volumetric heat sources or sinks. The later will often be modelled as step (switch) functions while the other will result from some long term data measurements. The set of tools provided for modelling periodic data includes for example: values at equidistant points (regular points), mean values at equally or irregularly distributed time intervals (regular means, irregular means), values in steps (switching) on irregularly tiled intervals (irregular steps). Furthermore any periodic distribution given by complex coefficients of Fourier series can be employed too.

Compared to [HARMONIC option](#) (this earlier option of AnTherm is available already for years) the TRANSIENT option adds higher precision of results, ability to evaluate higher harmonics of the main period requested (up to 1000 harmonics), evaluate timelines of temperatures at arbitrary probe points or visualize time dependant animations in 3D (by employing the already well known techniques of “standard” steady state AnTherm to chosen time series) and record them for movie reproduction.

The strength of the [HARMONIC tool](#) relies on the fact, that harmonic thermal conductances (complex number, amplitude and the phase lag) can be calculated without the need to enter nor to know any boundary conditions.

The TRANSIENT Option goes much further by adding the ability of modelling periodic boundary conditions (fully automated Fourier analysis) for the purpose of time dependant evaluations (either a 3D view at some point in time, then available to creating time series animation, or timelines of temperature changing in time at selected location of the 3D model).

See also: [Theoretical background](#), [European standards on thermal bridges](#), EN ISO 13786, EN ISO 13370

Computational performance enhancement for multiprocessor systems

Running solver on multiple CPUs enhances the calculation performance and thus (in most cases) reduces significantly the calculation time. [Multi CPU \(or Multicore\) behaviour of the solver is controlled by respective application settings](#) separately.

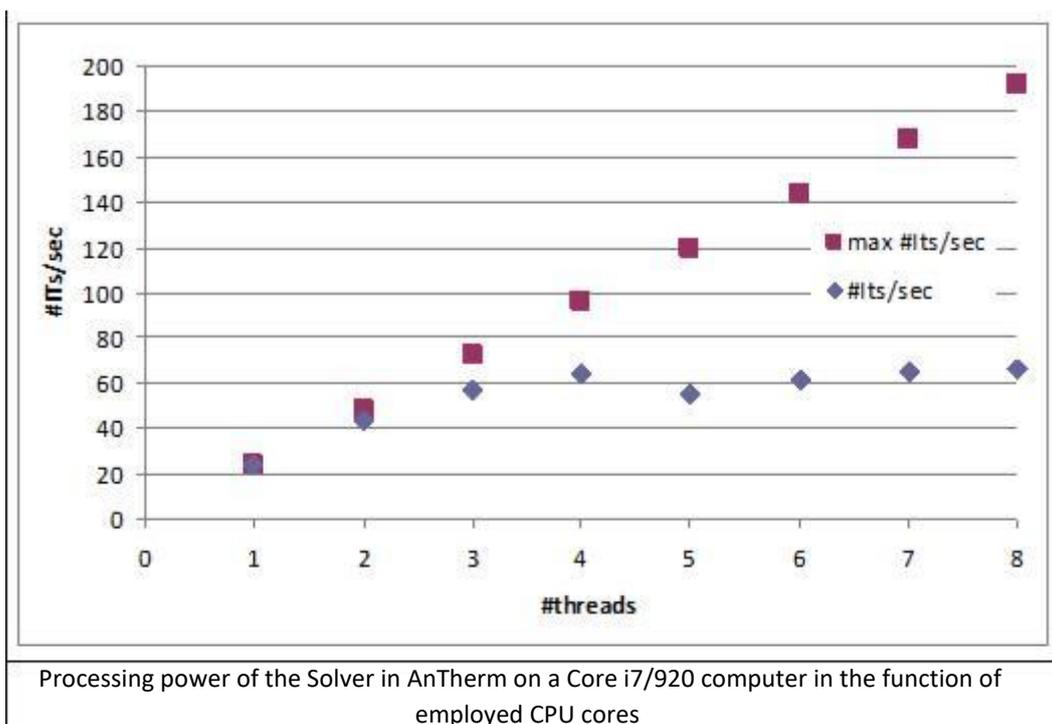
Remark: This function requires a valid license feature [MULTICORE](#).

MULTICORE-option: *Speeding up computationally intensive jobs by distributing them on multiple processors or processor cores for parallel execution is only possible with an active MULTICORE-Option of the program.*



Complex calculations (such of more than 1 million equations) can be critical in terms of time factor - the calculation can last few seconds only but (for so called "bad conditioned equations") tenths of minutes (on currently typical PC computer hardware). The CPU speed, CPU type but also die memory architecture of the system determine the effective computational power which is available to AnTherm.

By employing the MULTICORE-Option the software can access multiple CPUs oder CPU kernels and by that significantly improve the efficiency. By increasing the number of used kernels will add to the computational power as expected. Knowing that it depends on the operating efficiency of many other components of the device it is advisable to determine the optimal number of cores by executing simple measurement.



Above picture shows, that the theoretical multiplication of the computing power will not be reached by simply raising their number of cores! For the computer architecture used for the above example the saturation of computational power is reached already with four cores. This results from the technical structure of the Core i7/920 processor having only for memory paths (one for each pair of cores). In reality saturating the memory bandwidth raises the computing power limit in that case.

The number of cores used by the solver shall be set in the application settings.

See also: [Solver window](#), [Application settings](#)

Immersive 3D stereo (binocular) experience

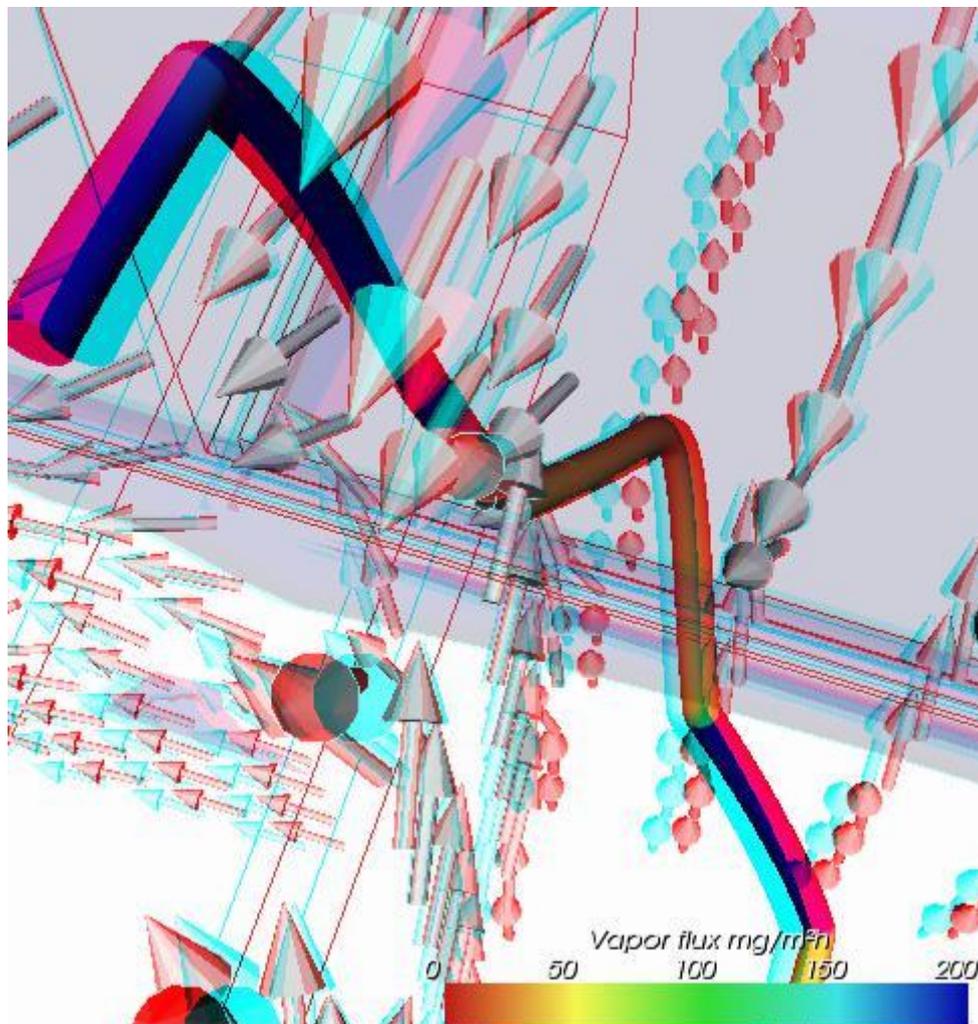
AnTherm utilizes stereo viewing capabilities for the three dimensional analysis of thermal bridges in building constructions. In addition to the real time 3D rendering of analyzed objects one can now easily receive immersive experience of real 3D by turning into stereo capable interactive imagery.

STEREO3D-option: Stereoscopic, binocular display of three dimensional evaluations is only possible with an active STEREO3DVIEW-Option of the program.



Stereo viewing is even possible with very cheap Read-Blue glasses. By utilizing more advanced equipment, such as Crystal-Eyes or NVIDIA 3D Vision glasses, fully immersive stereoscopic 3D experience of colourful visualizations can be made available to nearly any user's desktop PC.

By a simple click of the mouse AnTherm automatically transforms your PC into full stereoscopic 3D thermal bridge evaluation tool.



Remark: Using binocular stereo modes requires a valid license feature [STEREO3DVIEW](#)

See also: [3D Navigation within 3D windows](#)

Country specific localisation - important notice

While using international features, Windows provides you with country specific setting for decimal comma. This shall allow you to use numeric pad of your keyboard for entering numbers correctly. Please note, that entering 0,8 (with comma instead of dot) might result in 08 if your setting of current operating system's locale (typically Regional Setting of the System's Control Panel) defines different character (for example a dot) for decimal comma. The opposite might also be valid (e.g. German systems use comma - the , character) for decimal comma.

If the system is set up properly for your country, the numeric pad's comma will provide the right character to the input.

Note: The AnTherm® provided application [User interface language](#) setting does not affect the operating system setting for the decimal comma or dot.

See also: [Application settings](#)

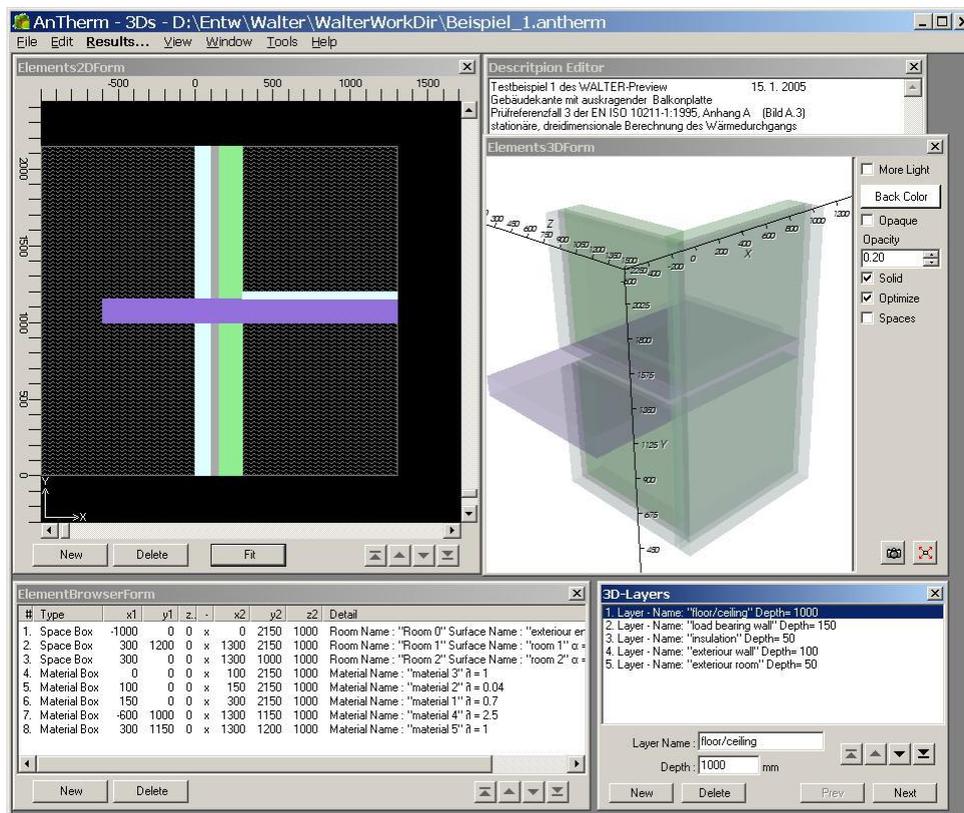
Starting the program

Start the application from *Windows Menu Start*→*Programs*→*AnTherm*→*AnTherm*.

It is also possible to start processing of a [AnTherm project file](#) directly from the Windows File Explorer. The double-click onto the project file ([AnTherm project file](#) has an extension .antherm) within the windows file explorer will start the program and open the chosen file.

If the [installation of the program](#) has been successfully completed, you will be shown the [Main application window](#) together with further application windows embedded in it. The [Tip of the Day](#) shown immediately after application start provides useful hints on application usage.

Pressing the **F1 key** or using the menu **Help**→**Contents** will show you the documentation - user guide, program reference, tutorials etc.



The windows of the program AnTherm will be shown on your screen at positions as they were shown on the last application use. On initial application start some preset windows are shown at their initially preset positions, depending on the [installed and loaded template](#). All windows can be freely moved and most also freely sized. This allows you to setup the layout and content of the screen according to your needs. During the final shutdown of AnTherm positions of windows are saved together with all further [application settings](#).

Depending on [application setting](#) AnTherm will on its startup either initially load a template project file `project.xml` from the installation folder or it will load the recently open project automatically.

Working with the program is controlled from the [main menu](#) and from various [editing, dialog](#) or [evaluation windows](#) and their context menus.

Tip: If immediately after starting the program the message „The application failed to initialize properly (0x000135). Click OK to terminate the application“ is shown, please check if „MS.NET Runtime“ is installed on your computer – see also [Installation of MS.NET Runtime](#) or [Installation of the Program](#) .

Tip: The program's *User Interface* is shown in the *language* chosen according to the preset setting of your operating system and language variant available in AnTherm (standard default language). You can change this! Please select the Menu Edit→Settings... (or Bearbeiten→Einstellungen...) and within the [settings dialog](#) set the [User Interface Language \(Sprache der Benutzeroberfläche\)](#) according to your needs from languages available in AnTherm.

VAPOUR-option: Analysis of multidimensional vapour diffusion is only possible with an active VAPOUR-Option of the program..



Tip: You will receive a warning if there is one instance of the application running already. There is only one instance of AnTherm allowed to be actively executed. Simultaneous execution of more than one instance will be blocked. When attempting to start second instance, such instance will be terminated and you will be shown informational message. However such behaviour can be changed with the [application setting "multiple app-instances allowed"](#). This functionality is implemented by acquiring the mutex "{4927A7F0-BD15-499d-A82A-5EE422C6FF3F}" (arbitrary chosen GUID).

See also: [main menu](#), [input windows](#), [main window](#), [MDI/SDI](#), [application settings](#), [standard project projekt.xml](#), [Installation of the program](#), [Tutorials und Cook-Books](#)

Project types

Following project types can be created and processed:

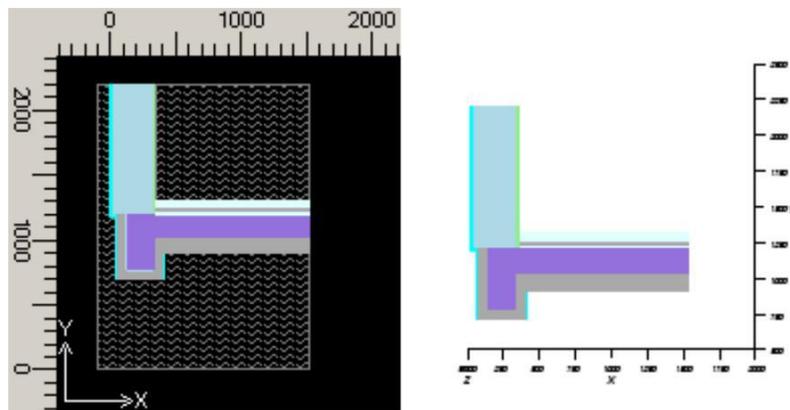
- [2D project](#)
- [Layered 3D project](#)
- [3D project](#)

Projects can be [converted to different type](#) too (e.g. using a 2D model as a template for more sophisticated 3D modelling).

2D project

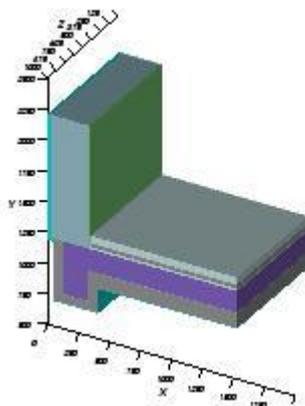
Primary property of a *2D project* is given by geometric modelling done in two dimensions only. Only one planar slice is entered and the resulting two dimensional construction calculated. There is no input regarding the third dimension - i.e. coordinates of the axis perpendicular to project plane.

This project type is typically used to model a linear thermal heat bridge.



Final 2D Projekt contains ordered sequence of elements. The order of elements is relevant, because later entered elements will [partly or fully overlap](#) elements earlier in the order if covering common plane area.

Note: Internally AnTherm represents the 2D model as a single [layered 3D project](#) with the thickness of exactly 1000 mm - i.e. 1,0 m - in Z direction.



This dependency shall be considered if, e.g. for some result display, two dimensional constructions are modelled in three dimensions.

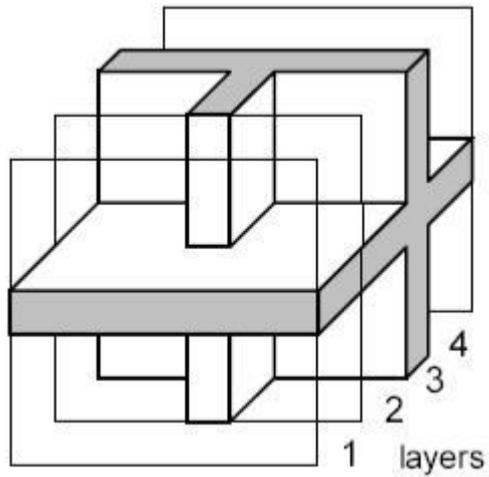
Remark: To support graphical entry the [underlay can be registered](#) and shown within the plane of [Elements2D window](#).

Remark: The 2D Project Type will be often uses as a starting point for [creating 3D models by various conversion capabilities](#) (see below).

Layered 3D project

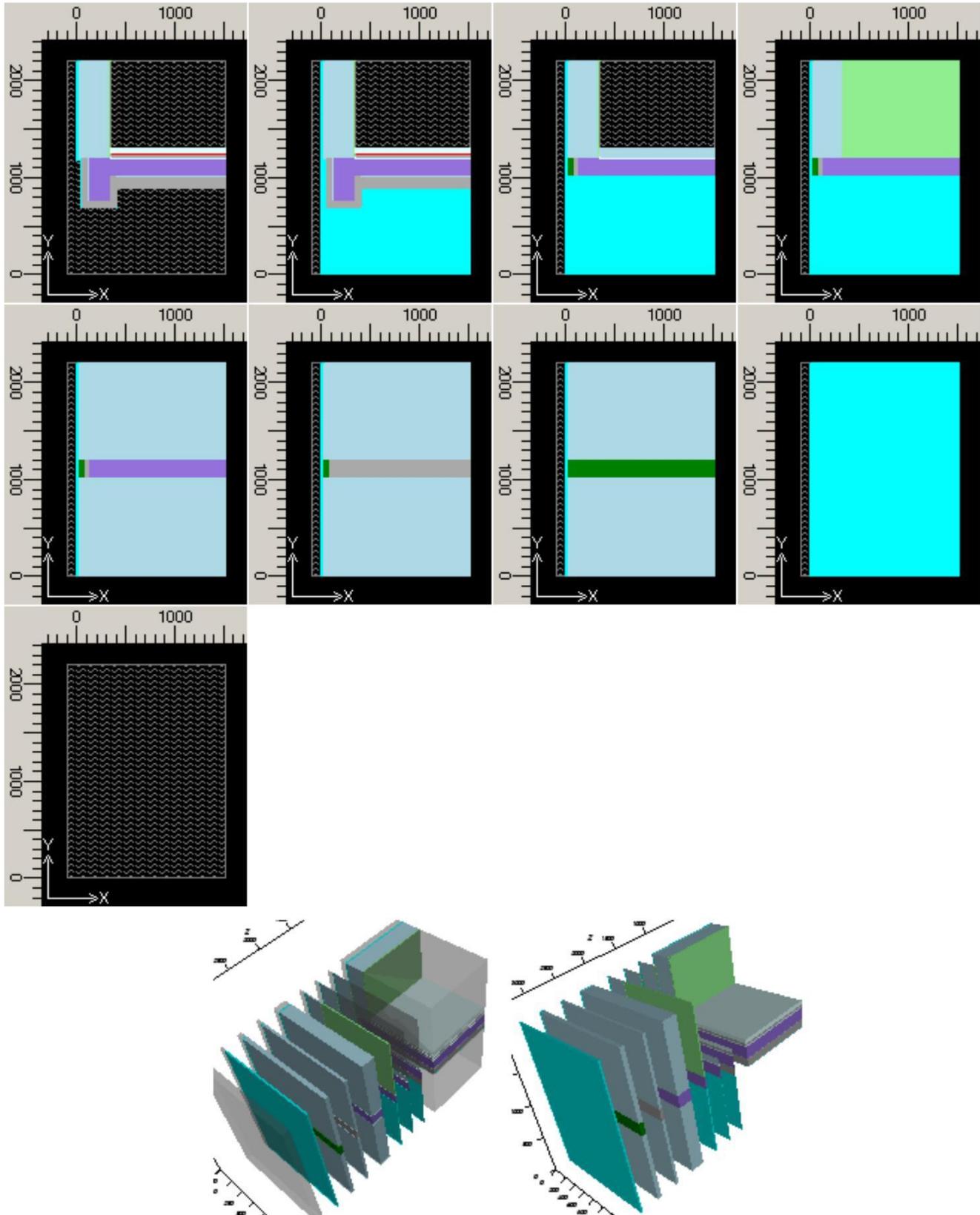
Primary property of a *layered 3D project* is given by its geometric modelling of planar slices consecutive to each other (slices itself are identical to a [2D project type](#)).

This project type is typically used to model a point thermal heat bridge.

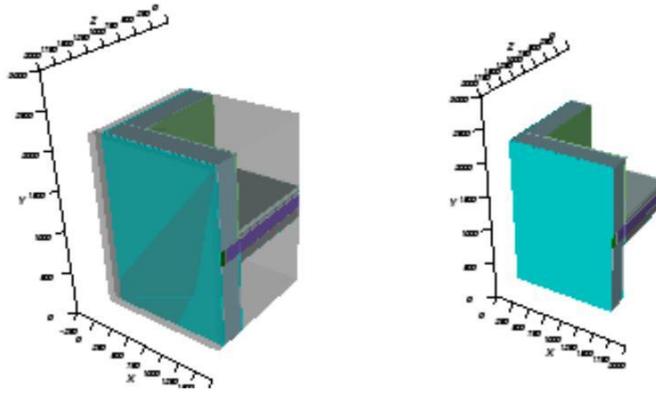


Three dimensional model is given by assigning a *thickness* to each such planar slice. In Antherm each such slice with assigned thickness is called *layer*.

Handbook Thermal Bridge Heat Transfer and Vapour Diffusion Simulation - AnTherm



Final "layered 3D project" contains ordered series of [layers](#). The sequence can be controlled by the [layers list](#).



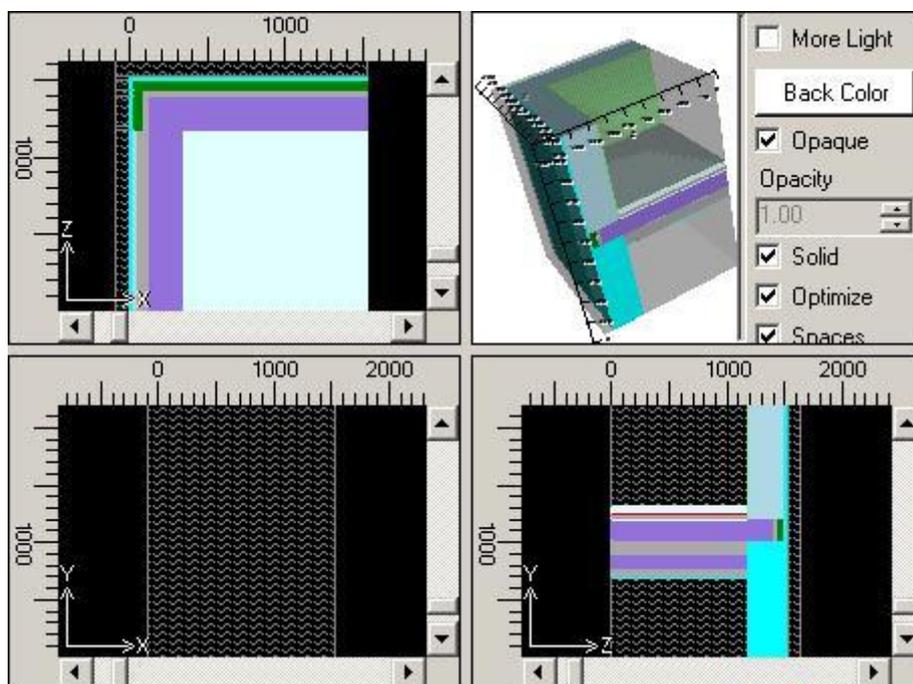
Note: A three dimensional model will result in exactly same results as its two dimensional counterpart only, and only if, the geometry and conductivities do not change along one dimension and the extent in this direction is exactly 1000 mm - i.e. 1,0 m.

Remark: To support graphical entry the [underlay can be registered](#) and shown within the plane of [Elements2D window](#).

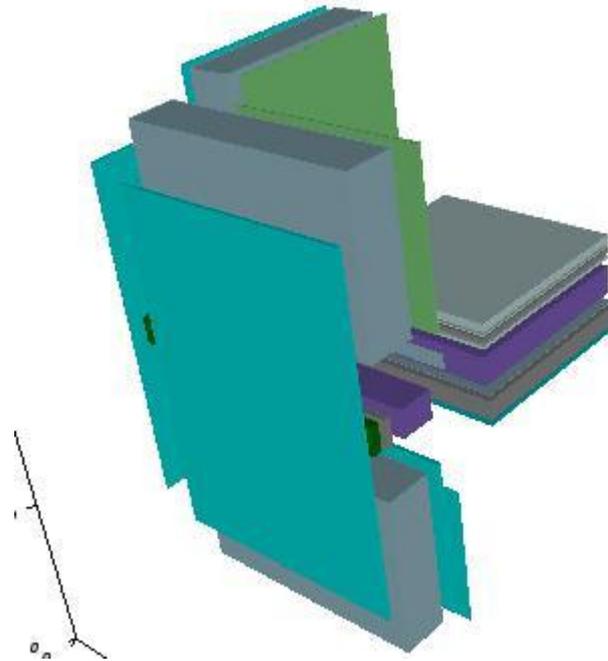
3D project

Primary property of a *3D project* is given by entering all three coordinates in three dimensions directly. The resulting geometry of building construction is described by series of cubes.

This project type is typically used to model a complex point thermal bridge.



The order of element cubes in the sequence is relevant, because later entered elements will [partly or fully overlap](#) elements earlier in the order if covering common space area.



Note: A three dimensional model will result in exactly same results as its two dimensional counterpart only, and only if, the geometry and conductivities do not change along one dimension and the extent in this direction is exactly 1000 mm - i.e. 1,0 m.

Remark: To support graphical entry three [underlays can be registered](#) and shown within respective planes of [Elements23 window](#) quadrants.

Converting between project types

- A [2D project](#) can be transformed (converted) to a [layered 3D project](#).
- A 2D project can be [revolved \(rotated\) to produce 3D project](#) based on a 2D template.
Warning: Revolution to a 3D project is irreversible.
- A [layered 3D project](#) can be converted to a [2D project](#) only if it contains only one layer.
- A [layered 3D project](#) (and a [2D project](#) too) can be converted to a [3D project](#).
- During the conversion AnTherm will assign each element to a additional group named „Layer#:LayerName/Thicknessmm“. This allows element selection by its original (prior to conversion) layer structure ([context menu of element processing->Select->By Group](#)).
Warning: Converting to a 3D project is irreversible.
- A [3D project](#) can be converted to a [2D project](#) only if thicknesses (e.g. Z coordinates) of all elements are equal. The resulting project will receive thickness of 1000mm (Z1=0, Z2=1000).
- A [3D project](#) can be converted to a [layered 3D project](#). The conversion will create 3D Layers in the Z-directions for all Z-coordinates existent in the project. Layer names will be derived from that Z-coordinate values. All elements are then added accordingly to respective layers.
Warning: This conversion need not be reversible.
Important: Element of Z-thickness equal 0mm will not be included in the conversion and ignored.

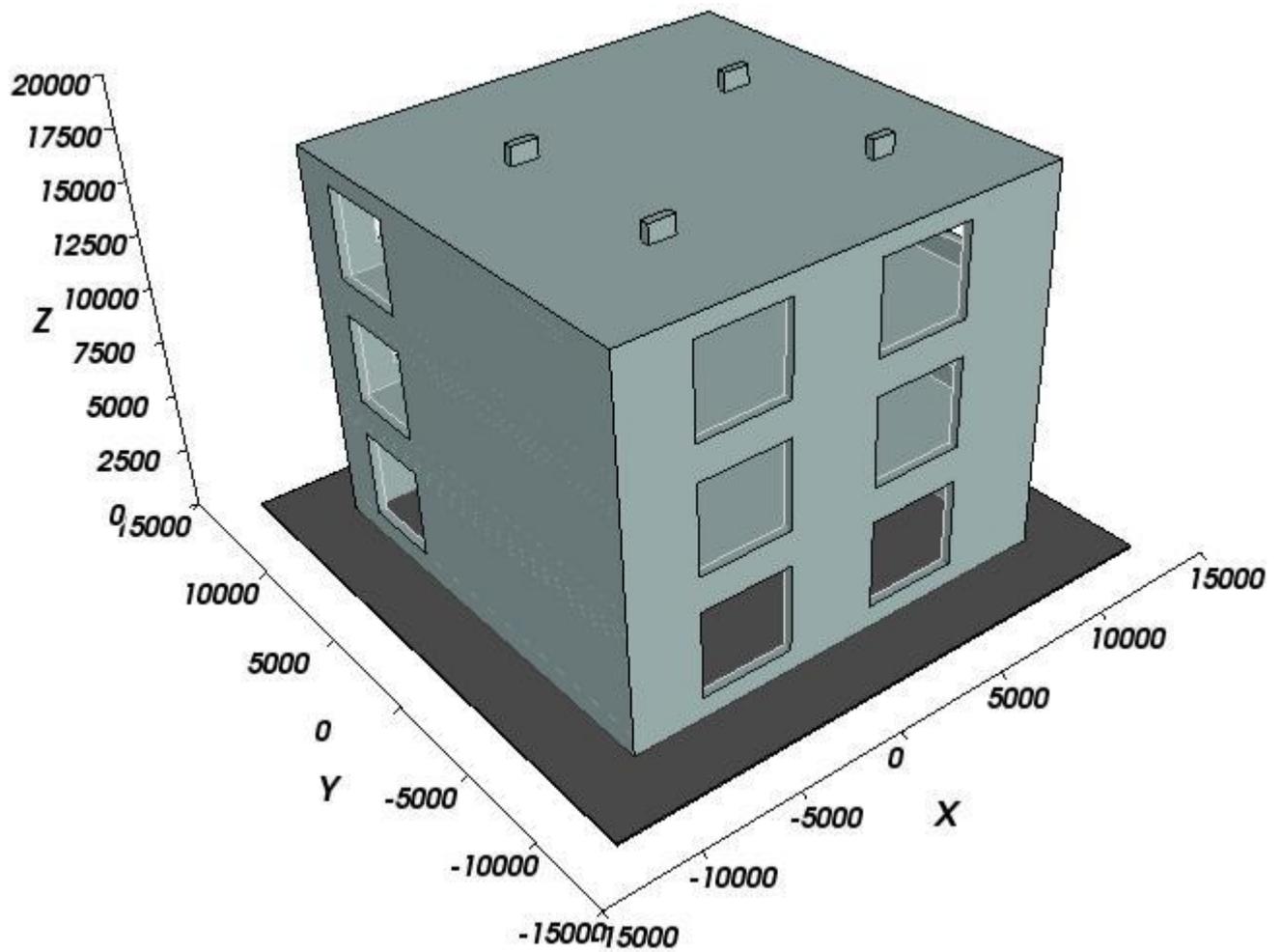
See also: [Create new project](#), [Converting a project](#), [Working with files](#), [3D-Layered model](#)

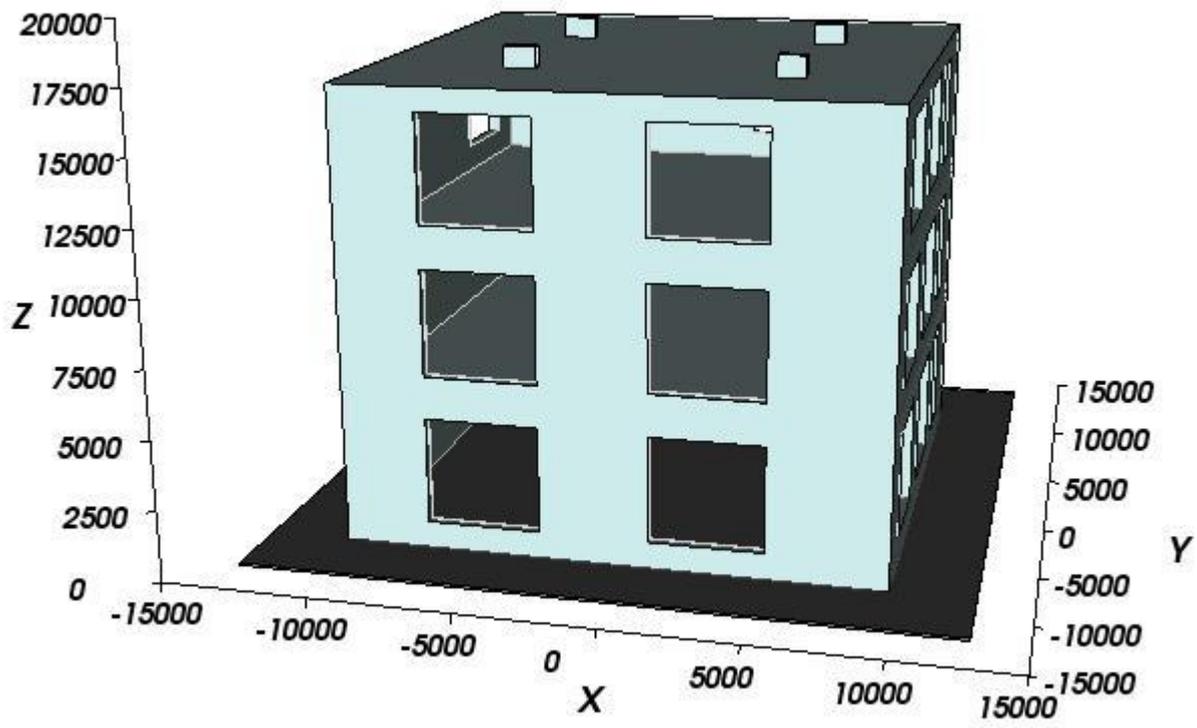
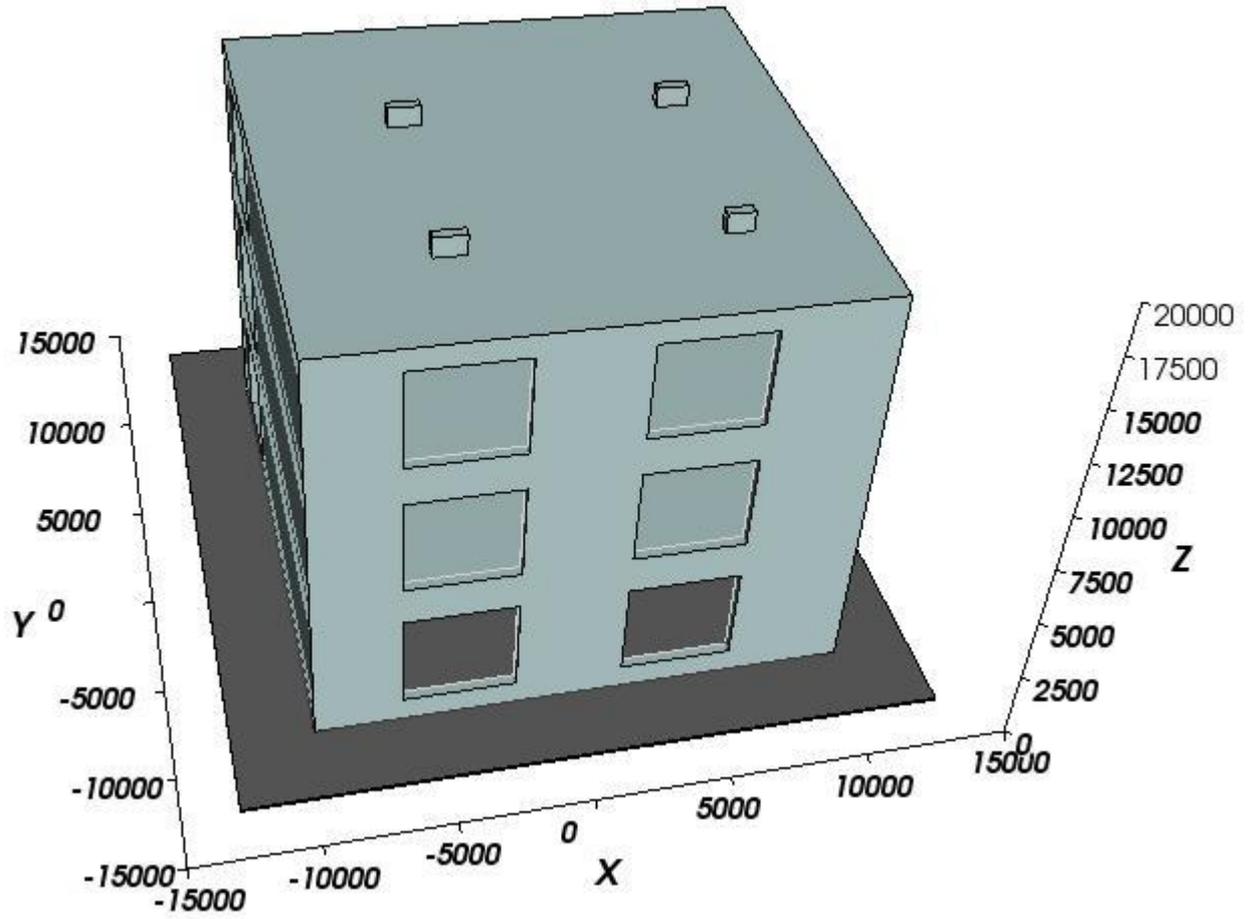
Coordinate system

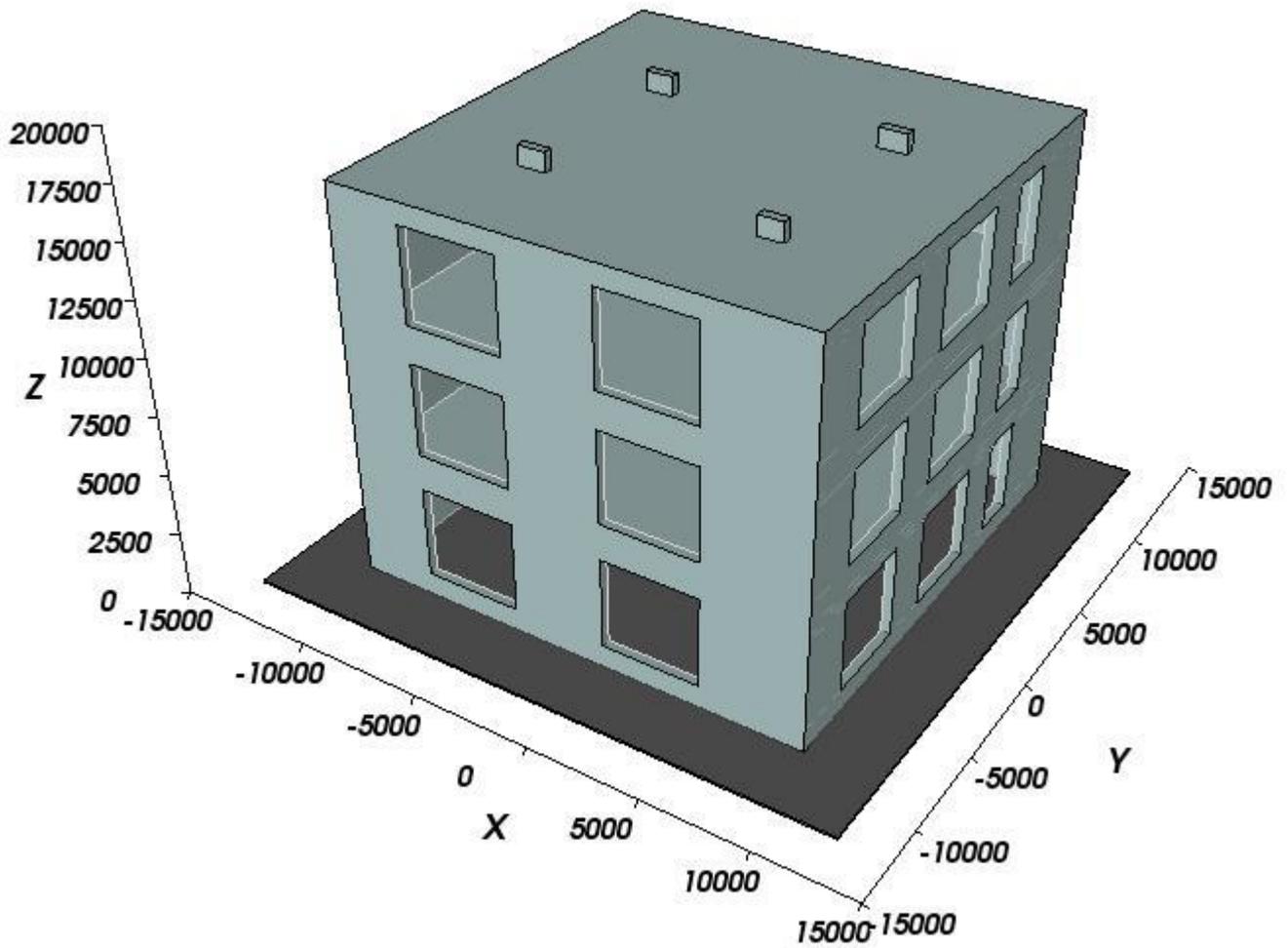
The three dimensional construction is being modelled (input) within a Cartesian coordinate system. Axes X, Y, Z are set up to create a positive oriented, orthogonal coordinate system.

Normally one places the footprint at the X-Y plane, thus coordinate values of the X axis grow to the right and those of the Y axis towards the back, hence the Z axis shows upwards along the structural levels.

The view from right is then parallel to the Y-Z plane, and the front view is parallel with X-Z plane.







The user can of course freely choose to place the footprint at any other plane. He shall merely observe the orientation of the coordinate system to avoid in advance any mistake or confusion regarding the direction of axes.

Note: Labelling of the axes titles can be modified in [application settings](#) .

See also: [Projec types](#), [Element Editor](#), [Elements 2D window](#), [Elements 3D window](#), [Elements 23 window](#)

Results & Evaluation

AnTherm provides variety of ways to show computational results and execute evaluations. It provides results as textual reports and graphical evaluations (visualization).

Reports

Reports provide computational results as textual, printable output. All reports can be

- viewed on the screen,
- saved as [PDF](#), [XLS](#), [DOC](#) or [RTF file](#) for further processing,
- printed on the printer and
- searched for specific words and phrases.

Following reports are available and can be requested from the [Main menu](#) of the program:

Report builder	The report generator allows you to create completely custom reports. Microsoft Word is required.
Data Entry Report	detailed listing of entered data
Modelling Report	a summarizing report for the modelled construction, used materials, spaces, surface properties and power sources
Coupling Coeff. Report	Output of the calculated matrix of Thermal Coupling Coefficients (the conductance matrix) along with calculation precision indicators.
Results Report	Output of surface temperatures under defined boundary conditions at coldest surface points of each space, along with their location (coordinates) and respective temperature factors f_{RSi} . The 2nd part of this report outputs temperature values at given probe points under defined boundary conditions . If there is vapour solution also, values of vapour saturation pressure and partial pressure are also shown. 

Each output form contains the name and address of the user displayed at the top. Date and time of the output creation along with the program information are also shown. Please refer to [license terms and conditions of use](#) regarding these output contents. Just below the heading the file name (the project file) is shown followed by the user chosen [short project description](#). [Specific](#) calculation and simulation results are displayed below this general information.

Graphical Evaluations (Visualization)

Graphical evaluations are used to visualize simulation results as two- and three-dimensional picture renderings. The choice of [primary function](#) provides following visualizations:

- Representation of [temperature distribution](#) (colour mapped picture, iso-lines/isotherms, iso-surface, slices, profile-diagram),
- Representation of [heat flux/heat stream flow](#) (colour mapped, isolines, streamline(s), vector arrow(s), isosurface, slices, profile-diagram) und
- Representation of [dew point](#) (colour mapped view, isolines),
- Representation of [interstitial \(core\) vapour pressure distribution](#) (partial pressure, saturation pressure or pressure difference, relative core humidity) (colour mapped view, isolines/isobars, isosurface, slices, profile-diagram)
- Representation of [vapour diffusion flux/stream flow](#) (colour mapped view, isolines, streamline(s), vector arrow(s), isosurface, slices, profile-diagram). 
- Visualizations can be easily adjusted to user's needs and according to specific requirements:
- Changing the [transparency/opacity](#) of visible figures
- Selection of [value intervals and function specific adjustment](#)
- Choice of different [slice planes](#) or probing points
- Selection of used [colouring tables](#) or by choosing single [colouring](#)
- [Switching to On/Off](#) of specific elements of visualisations (surface, edges, axes, slice planes etc.)
- Adjusting further display parameters. like [light effects](#), [position and size of the picture](#) etc.

At any time the graphical display can be [copied to the clipboard](#) for further transfer to other applications (for example to create compound reports).

Visualizations can also be saved (exported) to [picture files \(PNG, GIF, JPG, TIFF, ...\)](#) or to [3-D scene files \(VRML, OOGL, IV, ...\)](#) for further processing.

Following visualizations can be shown in [Results 3D window](#) :

Temperature distribution:

Surface temperatures	Displaying the colour mapped distribution of temperature on the surface of the construction. The representation can be adjusted by choosing different colour table which assigns colour to intervals of the temperature values.
Isotherms of surface temperatures	Displaying the distribution of temperature on the surface of the construction by isolines, i.e. lines of constant temperature . Isotherms can be coloured by applying the colour table which assigns colours to temperature values. The assignment of isolines to values of temperature can be controlled by user input.
Temperature distribution on on a slice plane	Displaying the colour mapped distribution of temperature on a slice plane cutting through the construction. The representation can be adjusted by choosing different colour table which assigns colour to intervals of the temperature values. The position of the slice plane - always parallel to axes planes - can be set by input of the coordinate at the intersection of the axis perpendicular to the slice plane.
Isotherms on a slice plane	Displaying the colour mapped distribution of temperature on a slice plane cutting through the construction by isotherms, i.e. lines of constant temperature . Isotherms can be coloured by applying the colour table which assigns colours to temperature values. The assignment of isolines to values of temperature can be controlled by user input.
Isosurface for one value of temperature	Displaying a surface of constant temperature (iso-surface) for one freely chosen value of temperature. The assignment of the colour of the surface to the represented temperature is applied by selecting colour table .
Value of temperature at one chosen point.	Displaying a numerical value of temperature at one freely chosen point of the constructions. The position of this probing point is set by positioning slice planes XY/YZ/ZX . The value shown corresponds to the point of intersection of those three planes.
Diagram of temperature along a profile line.	Displaying the X/Y-diagram of temperatures along the chosen profile line. The position of this profile line point is set by positioning slice planes XY/YZ/ZX . The diagram shown corresponds to values along the intersection of two of those three planes.

Distribution of heat flux (heat stream and heat stream density):

Heat stream density on construction's surface	Distribution of heat flux (modulus of heat stream density) on the surfaces of the construction is shown as a coloured picture. The assignment of colours to heat stream density values depends on the chosen colour table .
Isolines of heat stream density on the construction's surface	Display the distribution of the heat stream density on the surfaces of the construction shown as isolines , i.e. lines of constant heat stream density values. The colour of the lines depends on the colour table chosen.
Heat stream density on a slice plane	Displaying the colour mapped distribution of heat flux on a slice plane cutting through the construction. The colouring of the slice plane depends on the colour table chosen. The position of the slice plane is given by one coordinate.
Isolines of heat flux on a slice plane	Displaying isolines of heat stream density value on a slices plane cutting through the construction. Lines of constant stream density are coloured as set by the chosen colour table .
Isosurface for one value of heat stream density	Displaying a surface of constant value (iso-surface) for one freely chosen heat stream density. The colour of the surface depends on the colour table chosen.
Streamline(s) tracing the heat flow through the construction	Display of the heat flow streamline from/to any chosen point within the construction. The colour of the streamline depends on the colour table chosen.
Vectors (HedgeHog, Arrows) tracing the heat flow through the construction	Display of the heat flow at any chosen point or points within the construction by arrow glyphs. The size of vector symbols follows the heat stream density. The colour of glyph symbols depends on the colour table chosen.

Heat flux diagram along a profile line .	Display of a X/Y-diagram of heat stream density along the chosen profile line. The position of this profile line point is set by positioning slice planes XY/YZ/ZX . The diagram shown corresponds to values along the intersection of two of those three planes.
--	---

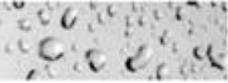
Dew point values at construction's surfaces

Dew point values at construction's surfaces	Displaying the colour mapped distribution of dew point values on the surface of the construction (calculated from the surface temperature and space air temperature). The representation can be adjusted by choosing different colour table which assigns colour to intervals of the dew point values.
Isolines of dew point value on construction's surfaces	Display of the distribution of dew point values on the surface of the construction shown as isolines (lines of constant dew point value). The assignment if the colours to dew point values depends on the chosen colour table .

Distribution of vapour pressure difference, saturation and partial pressure, relative interstitial core humidity

Pressure distribution on the construction's surface	Display of pressure or pressure difference distribution on the surfaces of the construction is shown as a coloured picture. The assignment of colours to pressure values depends on the chosen colour table .
Isolines of pressure distribution on the construction's surface	Display of pressure or pressure difference distribution on the surfaces of the construction is shown as isolines, i.e. lines of constant values . The colouring of isolines is applied according to the chosen colour table .
Pressure- or pressure difference distribution on a slice plane	Displaying the colour mapped distribution of pressure or pressure difference on a slice plane cutting through the construction. The representation can be adjusted by choosing different colour table which assigns colour to intervals of pressure values. The position of the slice plane - always parallel to axes planes - can be set by input of the coordinate at the intersection of the axis perpendicular to the slice plane.
Isolines of vapour pressure on a slice plane	Displaying isolines of vapour pressure or pressure difference on a slices plane cutting through the construction. Lines of constant pressure are coloured as set by the chosen colour table .
Isosurface for one pressure or pressure difference value	Displaying a surface of constant value (iso-surface) for one freely chosen vapour pressure or pressure difference. The colour of the surface depends on the colour table chosen.
Output of pressure difference at one chosen point .	Displaying a numerical value of pressure difference at one freely chosen point of the constructions. The position of this probing point is set by positioning slice planes XY/YZ/ZX . The value shown corresponds to the point of intersection of those three planes.
Pressure difference diagram along a profile line .	Display of a X/Y-diagram of pressure difference along the chosen profile line. The position of this profile line point is set by positioning slice planes XY/YZ/ZX . The diagram shown corresponds to values along the intersection of two of those three planes.

Distribution of the value of vapour stream density

<p>VAPOUR-Option: <i>The analysis of multi-dimensional water vapor diffusion processes is only possible with the Vapour option of the program.</i></p> 	
Distribution of vapour stream density on construction's surface	Display of vapour stream density distribution on the surfaces of the construction is shown as a coloured picture. The assignment of colours to density values depends on the chosen colour table .
Isolines of vapour stream density on construction's surface	Display of vapour stream density distribution on the surfaces of the construction is shown as isolines, i.e. lines of constant values . The colouring of isolines is applied according to the chosen colour table .
Vapour stream density on a slice plane	Displaying the colour mapped distribution of vapour stream density on a slice plane cutting through the construction. The representation can be adjusted by choosing different colour table which assigns colour to intervals of density values. The position of the slice plane - always parallel to axes planes - can be set by input of the coordinate at the intersection of the axis perpendicular to the slice plane.
Isolines of vapour stream density on a slice plane	Displaying isolines of vapour stream density on a slices plane cutting through the construction. Lines of constant pressure are coloured as set by the chosen colour table .
Isosurface for one vapour stream density value	Displaying a surface of constant value (iso-surface) for one freely chosen vapour stream density. The colour of the surface depends on the colour table chosen.
Streamline(s) tracing the vapour diffusion flow through the construction	Display of the diffusion flow streamline from/to any chosen point within the construction. The colour of the streamline depends on the colour table chosen.
Vectors (HedgeHog, Arrows) tracing vapour diffusion flow through the construction	Display of the diffusion flow at any chosen point or points within the construction by arrow glyphs. The size of vector symbols follows the vapour stream density. The colour of glyph symbols depends on the colour table chosen.
Vapour flux diagram along a profile line .	Display of a X/Y-diagram of vapour stream density along the chosen profile line. The position of this profile line point is set by positioning slice planes XY/YZ/ZX . The diagram shown corresponds to values along the intersection of two of those three planes.

See also: [Evaluation windows](#), [Results 3D window](#), [Main menu](#), [Main window](#), [Secondary functions](#), [Active function](#), [Toolbar of Report windows](#)

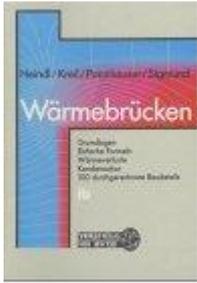
Basics and Some Theory of AnTherm

Primary Concepts

The various terms and theoretical concepts, which form the basis for understanding the *AnTherm* application, are presented and defined briefly in the following sections.

For a more thorough explanation of the theoretical background, please refer to the

book "[Wärmebrücken](#)" Read more on theoretical concepts of *AnTherm*:



See also: [Theoretical background](#)

Basics and Some Theory of AnTherm

Environmental Factors

When analysing the thermal performance of building components, two different types of information are sought: thermal transmission and surface temperatures.

The first type entails evaluating the heat transfer characteristics of a building assembly, i.e. the form and quantity of heat losses and gains, in order to ultimately obtain

- a reliable estimation of heating and cooling loads (costs),
- acceptable heat flow rates,

as well as

- energy conserving design alternatives.

Based on the heat transfer characteristics of a construction, the expected temperatures along interior surfaces must be evaluated in order to predict (and avoid) areas of potential moisture condensation. Beyond preventing damage to building materials caused by mould growth, adequate surface temperatures are also a relevant factor in the thermal comfort of an interior environment.

See also: [Thermal Transmission](#), Surface Temperatures

Basics and Some Theory of AnTherm

Thermal Transmission and Thermal Bridges

Energy in the form of heat in building constructions is generally transmitted by a combination of conduction, convection, and radiation. When considering heat transfer within and through solid (homogeneous or non-homogeneous) building materials, heat conduction is the primary transmission factor; the effects of convection and radiation are typically negligible.

heat conduction Conduction through homogeneous planar building components (i.e. composed of one or more layers of material with parallel surface planes) occurs in a single direction: normal to the component surface. This is referred to as *one-dimensional heat flow*, and is characterised by a constant surface temperature over the entire surface plane. Such idealised conditions can, of course, only be assumed in limited regions of an actual building structure. Geometries of non-planar components (construction joints, floor-wall connections, balconies, etc.) give rise to heat flow patterns of more than one direction, that is, to two- or three-dimensional heat flow.

thermal bridge
cold bridge
heat bridge Regions or elements of a building construction characterised by multi-dimensional heat flow patterns are called *thermal bridges*. In contrast to regions of one-dimensional heat flow, thermal bridges are typically associated with local peaks of heat loss, which correspond to characteristic drops in the interior surface temperature.

Thermal bridges, which in general occur at any junction between building components or where the building structure changes composition, have two consequences:

- a. a change in heat flow rate
and
- b. a change in internal surface temperature

compared with those of the unbridged structure.

Although similar calculation procedures are used, the procedures are not identical for the calculation of heat flows and of surface temperatures.

thermal bridge (defined in ISO 10211) Part of the building envelope where the otherwise uniform thermal resistance is significantly changed by:

- full or partial penetration of the building envelope by materials with a different thermal conductivity and/or
- a change in thickness of the fabric and/or
- a difference between internal and external areas, such as occur at wall/floor/ceiling junctions

linear thermal bridge Thermal bridge with a uniform cross-section along one of the three orthogonal axes (as defined by ISO 10211)

point thermal bridge Localized thermal bridge whose influence can be represented by a [point thermal transmittance](#) (as defined by ISO 10211)

See also: [Thermal Heat Conductance](#), [Methods of thermal heat bridge survey in international perspective](#)

Basics and Some Theory of AnTherm Conductance

An exact description of the thermal behaviour of a building assembly would by nature be non-linear, and thus very complicated to evaluate. Fortunately, for most construction situations of interest, it is possible to drastically reduce the complexity of the physical model without sacrificing any appreciable accuracy.

A linear description of thermal transmission involving an entire building can be introduced by

- assuming that the temperature of each environment (space) associated with the building is unique, i.e. independent of position, and
- forfeiting an exact treatment of radiation exchange within the spaces in favour of an approximation of this factor through adjusted surface transfer coefficients.

thermal conductance, L^{3D} In light of this simplified physical model, it can be stated that the amount of heat which flows from one space to another, Q , is proportional to the temperature difference between the two environments under consideration (i and j). The factor of proportionality is a conductance, L_{ij} , and linearly defined by

[W/K]
$$Q = L_{ij} \cdot (T_i - T_j)$$

conductance matrix The analogy of an electrical circuit can be extended to model a building structure more generally as a set of environments thermally connected through a resistance, i.e. the heat-conducting material elements. A complete description of the thermal relationships in a particular structure is given by a conductance matrix of values for all thermal connections, $i \cdot j$, between the spaces of the model.

The conductance matrix defines the heat transfer characteristics of a given model based solely on the geometry and materials (thermal properties) of the resistance, that is, independently of temperature conditions in adjacent spaces. This matrix is symmetrical.

thermal transmittance, U -value, L^{1D} The area-related term "thermal transmittance" (U -value in $\text{Wm}^{-2}\text{K}^{-1}$) commonly used in standards to date essentially describes the same conductance as the reciprocal of the sum of resistances of a planar component, that is, resistances in "series":

[W/m²K]
$$1/U = R = R_{si} + \sum R_j + 1/\alpha_{se} = 1/\alpha_{si} + \sum R_j + 1/\alpha_{se}$$

whereby α_{si} and α_{se} are the surface transfer coefficients of the interior and exterior environments, and $\sum R_j$ represents the sum of material resistances of j constituent component layers. The resistance of an individual homogeneous (isotropic) layer is directly proportional to layer thickness, d , and indirectly proportional to the material conductivity, λ :

$$R_j = d/\lambda$$

However, this simple formula applies only to the planar regions of a building assembly in which strictly one-dimensional heat flow can be reasonably assumed.

length-related conductance, L^{2D} A further type of special situation is given where two-dimensional heat flow patterns are to be expected. Such a region is a stretch of the building assembly which can be evaluated with respect to a two-dimensional section - under the assumption that no heat flow occurs normal to the section plane. In this case, a length-related conductance, L^{2D} [$\text{Wm}^{-1}\text{K}^{-1}$], must be calculated for the applicable region.

[W/mK]

The most general case, of course, is that of three-dimensional heat flow. For the regions of a construction in which no directional assumptions can be made about local heat flow patterns, only the evaluation of the (3D) conductance, L^{3D} [WK^{-1}], provides a reliable indicator of the thermal behaviour.

**total
conductance**

Due to the linearity of conductances, an entire building can be modelled as a sum of parts, with each part evaluated according to the applicable geometric conditions. Of course, the summation of conductances is only applicable if the temperature difference is the same through all model parts (e.g. one interior and one exterior temperature valid for the whole building).

The model is sub-divided by introducing theoretical cut-off planes, which must be located such that any potential heat flow through these planes can be considered negligible.

The total thermal conductance of a building thus modelled can be written as

$$L = \sum_i L_i^{3D} + \sum_j L_j^{2D} \cdot l_j + \sum_k U_k \cdot A_k$$

whereby l_j is the length over which the two-dimensional conductance, L2D, is valid for part j , and A_k is the area of validity for U_k .

This reliable and flexible approach to analysing the thermal performance of buildings is referred to in the Standards as the direct method. It requires the implementation of a suitable computer program for attaining two- and three-dimensional conductance results with the necessary precision

The program AnTherm makes use of the linear nature of the heat conduction model by first determining a generally applicable calculation model: a characteristic set of temperature-independent base solutions (see also "Method of Analysis").

See also: [Linear and Point Transmittance](#), [The building envelope as thermal heat bridge](#), [Theoretical background](#)

Basics and Some Theory of AnTherm

Linear and Point Thermal Transmittance

Often thermal bridges will be described in terms of linear and point thermal transmittances characteristic to building components analysed.

linear thermal transmittance, Ψ heat flow rate in the steady state divided by length and by the temperature difference between the environments on either side of a thermal bridge (definition in ISO 10211)
NOTE The linear thermal transmittance is used as a correction term for the linear influence of a thermal bridge.
[W/mK]

Psi The linear thermal transmittance is given by:

$$\Psi = L^{2D} - \sum_{j=1}^J U_j \cdot b_j$$

where:

- Ψ is the linear thermal transmittance *Psi* of the linear thermal bridge separating the two environments being considered;
- L^{2D} is the thermal coupling coefficient obtained from a 2-D calculation of the component separating the two environments being considered;
- U_j is the thermal transmittance of the 1-D component *j* separating the two environments being considered;
- b_j is the length within the 2-D geometrical model over which the value U_j applies
- J is the number of 1-D components.

Note: When determining the linear thermal transmittance, it is necessary to state which dimensions (e.g. internal or external) are being used because for several types of thermal bridges the value of the linear thermal transmittance depends on this choice.

point thermal transmittance, χ heat flow rate in the steady state divided by the temperature difference between the environments on either side of a thermal bridge (definition in ISO 10211)
NOTE The point thermal transmittance is used as a correction term for the influence of a point thermal bridge.
[W/K]

chi The point thermal transmittance is given by:

$$\chi = L^{3D} - \sum_{i=1}^I U_i \cdot A_i - \sum_{j=1}^J \Psi_j \cdot l_j$$

where:

- χ is the point thermal transmittance *Chi* of the point thermal bridge separating the two environments being considered;
- L^{3D} is the thermal coupling coefficient obtained from a 3-D calculation of the 3-D component separating the two environments being considered;
- U_j is the thermal transmittance of the 1-D component *j* separating the two environments being considered;
- A_j is the area over which the value U_j applies;
- Ψ_j are respective linear thermal transmittances (see above);
- l_j is the length over which the value Ψ_j applies;
- J is the number of 2-D components.
- I is the number of 1-D components.

Note: When determining Ψ and χ values, it is necessary to state which dimensions (e.g. internal or external) are being used because for several types of thermal bridges the Ψ and χ values depend on this choice..

Rewriting the above equation by replacing the linear thermal transmittance by its definition, provides following alternative means of calculating the point thermal transmittance:

$$\chi = L^{3D} + \sum_{i=1}^I U_i \cdot A_i - \sum_{j=1}^J L_j^{2D} \cdot l_j$$

An alternative expression for the total coupling coefficient L_{ij} which uses the linear and point thermal transmittances, Ψ and χ , is then given by

$$L_{ij} = \sum_{k=1}^K U_{k(i,j)} \cdot A_k + \sum_{m=1}^M \Psi_{m(i,j)} \cdot l_m + \sum_{n=1}^N \chi_{n(i,j)}$$

where:

- $U_{k(i,j)}$ is the thermal transmittance of part k of the room or building;
- A_k is the area over which the value $U_{k(i,j)}$ applies;
- $\Psi_{m(i,j)}$ is the linear thermal transmittance *Psi* of part m of the room or building;
- l_m is the length over which the value $\Psi_{m(i,j)}$ applies;
- $\chi_{n(i,j)}$ is the point thermal transmittance *Chi* of part n of the room or building;
- K is the number of thermal transmittances.
- M is the number of linear thermal transmittances;
- N is the number of point thermal transmittances;

Note: In above formula $\sum A_k$ is equal to the total surface area of the envelope.

Note: L_{ij} is equivalent to the heat transfer coefficient, H often used in other standards.

See also: [Conductance](#) , [On calculation of \$\Psi\$ -values for building constructions in connection with ground, Theoretical background](#)

Basics and Some Theory of AnTherm Surface Temperatures

Once the base solutions describing a particular model have been established, temperature (boundary) conditions can be applied in the adjacent environments in order to predict a specific temperature distribution in the building component.

condensing humidity / dew point Here the temperature along the component surface - where condensation is most likely to occur - is of particular interest. Humidity exceeding the saturation level of an environment, i.e. the [condensing humidity or dew point](#) [%], leads to condensation. This value is directly linked to the temperature at any given location under.

temperature extremes Therefore [surface points at which a temperature minimum can be expected to occur must be localised](#) in the model and explicitly evaluated with respect to specified conditions:

- boundary conditions - air temperatures [°C]
- position of coldest surface point (x, y, and z coordinates)
- temperature of the surface at this point [°C]
- associated condensing humidity (the dew point) [%]

Note: Temperature maxima, i.e. warmest surface points, are primarily of interest when assessing the factor of thermal comfort in constructions which include heat sources. For example, situations in which electrical heating could be implemented (as floor heating, or as a measure against condensation) may be evaluated comparatively to determine if the power output needed could lead to uncomfortably high surface temperatures.

Building elements which absorb and re-emanate solar gain can also be effectively approximated as heat sources and thus evaluated in order to assess constructions critically affected by solar radiation.

Condensation Besides the avoidance of harmful water vapour [condensation the mould growth or corrosion](#) are also of importance.

Mould growth The mould growth can hold at 80% RH already.

Corrosion/Rust

Corroding events can start even at 60% RH.

Basics and Some Theory of AnTherm Weighting

The air temperatures set in the spaces adjacent to a building component all contribute to the temperature distribution resulting within the component. Therefore, the specific temperature, T , at any given point in a model can also be described as the result of all the space temperatures, T_0 through T_n , weighted for the specific position and summed:

$$T = g_0 \cdot T_0 + g_1 \cdot T_1 + \dots + g_n \cdot T_n$$

g - The set of weighting factors, g_0 through g_n , must be determined for each point of the model to be considered more *values* closely. These so-called *g-values* are normalised such that their sum is equal to one.

If all the interior spaces are set at the same temperature (T_i for T_1 through T_n), and T_0 is defined as the exterior temperature (T_e), then the equation above can be simplified to

$$T = g_0 \cdot T_e + (1 - g_0) \cdot T_i$$

and re-written as

$$T = T_i - g_0 \cdot (T_i - T_e)$$

In this case, g_0 represents a generalised version of the *f-value* familiar from the evaluation of surface temperatures for one-dimensional heat flow (based on *U-values*). Contrary to a conventional *f-value*, which applies to the entire interior surface plane, the *g-value* above is only valid for a specific point of a thermal bridge.

Once *g-values* have been characterised for the coldest points of all interior surfaces, however, the temperatures resulting at these points can be evaluated as simply as with the one-dimensional method **Factors**.

Basics and Some Theory of AnTherm

Graphic Representation of Heat Flow

Since the heat flow pattern of a thermal bridge is characteristically more complex than the single-direction flow through a planar component, graphic illustrations of bridge flow patterns can provide critical information at a glance.

heat flow diagram One method of visualising two dimensional heat flow is to [delineate the direction of flow \(vector\) through the building component with lines](#) drawn at prescribed intervals. The area bounded by two lines in the diagram represents a heat quantity defined by the interval (e.g. given 10 intervals, the heat flow between two lines corresponds to 10% of the total heat flow in one direction through the surface of a given space). The denser the lines in a region of such a diagram, the more heat flows through this region. Thus local peaks of heat loss (and predictably cold surface points) are easily discernible.

Due to the two-dimensionality of graphic illustrations, however, three-dimensional heat flow patterns cannot be meaningfully rendered with such a diagram. Therefore the three-dimensional rendering provides one streamline through the arbitrarily chosen point.

isotherms The second common method of heat flow representation is to render the temperature distribution in a component with [isotherms](#), i.e. by delineating lines of the same temperature at defined intervals. Isotherms lie normal to the direction of heat flow, thus providing an "inverted" rendering of the heat flow pattern (denser isotherms correspond to regions of increased heat flow).

Since isotherms represent a two-dimensional section through a temperature distribution rather than vectors directly, the isotherm method is also suited for (partial) rendering of three-dimensional heat flow situations (on surfaces or slices).

heat flux diagram Further method of heat flow representation is to render the [heat flux distribution](#) in a component - e.g. as [colorized](#) representation of values on [surfaces](#) or [slices](#) or with [isolines](#) of the same heat flow density. Such rendering provides clear heat flow pattern and allows easy identification of regions of increased heat flow.

isosurface For the three dimensional case the use of [isosurface](#) provides valuable alternative to isolines.

As for isolines the isosurface is useful to emphasize a surface of same temperature and to assess the irregularities of temperature distribution within the components body in 3D. The isosurface of the temperature field lies normal to the direction of heat flow, thus providing an "inverted" rendering of the heat flow pattern.

By utilizing the view onto the isosurface of equal values of heat flux one can visualize areas of increased heat flow rates and by that easily identify design, geometrical or material weak spots of the building component in three dimensional renderings.

Basics and Some Theory of AnTherm Method of Analysis

Heat flow in a building construction, as well as the subsequent temperature distribution, can be described mathematically by differential equations. Most importantly, these equations are linear and homogenous by nature.

base solutions This means that one set of solutions, calculated for a specific set of conditions, can be "[re-used](#)" as the basis for solutions under differing conditions by *superposition*, i.e. by linear combination of selected *base solutions*. More specifically, base solutions are calculated under the assumption of a "basic" set of boundary conditions: with an air temperature of 1 in the selected space, and 0 in all others.

Thus the temperature distribution, a function of the three spatial coordinates, can ultimately be written in the simple form of a sum of temperature values. The individual temperatures are the result of the actual boundary conditions (air temperatures in the given spaces from 0 to m) weighted by dimensionless base solutions:

$$U(x,y,z) = \sum_{j=0}^m T_j \cdot g_j(x,y,z)$$

In other words, base solutions are effectively a generalised form of weighting factors (g-values) - a function of position for a given space, j: $g_j(x,y,z)$.

The calculation approach in the program AnTherm uses this circumstance to minimise over-all evaluation time. One set of base solutions need be calculated only once to characterise a given model, which can subsequently be considered under varying boundary conditions without repeating the time-consuming computation necessary to solve the primary set of differential equations.

Computation time is further reduced by utilising the weighting function character of base solutions (normalised such that their sum must equal 1). Hence, if n cases have been selected, only n-1 solutions need to actually be calculated. The n-th base solution is then very simply derived as a difference of the sum to 1, that is, by a separate stage of superposition.

Basics and Some Theory of AnTherm Calculation Model

Using the concept of a conductance, L_{ij} (between the spaces i and j), the heat which flows out of space i is a sum depending on the temperatures of space i and of all other spaces, j :

$$Q_i = \sum_j L_{ij} \cdot (T_i - T_j)$$

Since L_{ij} can be proven to be symmetrical ($= L_{ji}$), this equation corresponds formally to Kirchhoff's law of electrical networks. The analogy that this allows is quite significant: the spaces of a building construction (model) can be thought of as nodes in a heat conducting network.

equation model Thus the heat conduction model, otherwise only describable with partial differential equations, can be mathematically defined as a relatively simple, linear system of equations with a given conductance matrix. Furthermore, only the thermal coupling between spaces which interchange heat directly through the building component ("neighbouring spaces") is of practical effect; coupling between indirectly connected spaces is generally negligible.

boundary conditions As the equation above shows, in addition to the appropriate value in the conductance matrix, either air temperature or heating load must be defined for each space of the model ([evaluation boundary conditions](#)) in order to attain a complete description of a particular heat flow situation.

Basics and Some Theory of AnTherm Numerical Solution

The calculation method of finite differences used by *AnTherm* is based on the sub-division of the heat conducting continuum into an orthogonal cell structure. Each cell of this calculation geometry is treated as a node of the thermal network, and is typically connected with six other such nodes. The conductance of each connection depends on cell size and thermal conductivity of the material "filling" the individual cells. Therefore, the cell structure must be defined such that each cell can be assigned a single, homogeneous material.

calculation geometry The calculation geometry is obtained by means of gridding the construction geometry of the model to be evaluated. A concrete grid is generated by dividing the model with a series of grid planes along material boundaries and at additional intervals (their number and distance depends on the precision of analysis required) parallel to the coordinate axes.

Once the network data has been sufficiently defined for calculation, the program sets up an appropriate linear system of equations automatically. This equation system then needs to be solved numerically with an iterative method.

relaxation factor The convergence process of the relaxation method used by *AnTherm* is entirely dependant on the value of the relaxation factor ω . Initially this factor is defined only within a given range:

$$1 \leq \omega < 2$$

The "optimal" ω -value is defined as the constant value of ω which leads to the most expedient convergence of the calculation method. This value differs from case to case and is initially unknown.

Once the system of equations describing the model has been determined, the first stage of calculation is therefore an analysis of this system, followed by an iterative calculation to approximate a preliminary "optimal" ω -value, ω_0 . This stage can be influenced by two parameters defined in the solver-parameters form:

- The termination condition for determining ω_0 is satisfied when the absolute difference between last value of ω_0 just calculated and the "old" value from the previous iterative step falls below a limit for which ω_0 is considered solved. This limit is called OMEGAO_DELTA within the solver-parameter set.
- A further termination condition is given by the maximum number of iterative steps which the program allows before stopping calculation and accepting the last value of ω_0 as optimal for the next stage of calculation. This step number limit is defined by OMEGAO_STOP.

A more precise approximation of ω_0 (e.g. by setting a more stringent termination condition or increasing calculation time) is usually unnecessary, since ω_0 just serves as an initial parameter for the relaxation factor ω , which is then modified with each iteration in the course of the second stage of calculation.

relaxation factor variation The range of ω variation,

$$\omega_{\min} \leq \omega < \omega_{\max}$$

depends on ω_0 for the lower limit according to the following equation:

$$\omega_{\min} = \omega_0 - k \cdot (\omega_0 - 1) \cdot (\omega_0 - 2),$$

whereby k is a pre-defined parameter between -1 and 0.

Since k is negative ($\omega_{\min} < \omega_0$), specifying a greater absolute value of k results in a larger difference between ω_{\min} and ω_0 , and thus a larger range. This parameter is set as OMEGA_MIN. The upper limit of the range of ω variation, ω_{\max} , is a set value between ω_0 and 2. This is defined as OMEGA_MAX.

In the second stage of calculation, the first step is performed with $\omega = \omega_{\min}$. The relaxation factor is then increased incrementally with each iteration until either ω_{\max} is reached or the process begins to diverge,

whereupon ω is set back to ω_{\min} and calculation continues.

The increments of ω variation also vary: low values of ω in the beginning are run through rapidly, but the increment approaches 0 asymptotically as ω approaches the value of 2. The increments are calculated such that the approximated optimal value of ω_0 is reached within a given number of iterations starting from ω_{\min} . The standard number of assumed steps is relatively small number set as OMEGA_WEIGHT=-3.

As already mentioned, ω is set back to ω_{\min} as soon as the iterative results begin to recognisably diverge. The criterium for discerning whether the solution process is converging or diverging is the absolute value, Δ_{\max} , i.e. the deviation of the results from one iteration to the next. Hereby a mean value of Δ_{\max} is calculated from the last n steps and compared continually with the current Δ_{\max} . The number of steps included in the mean value for comparison, n , is also defined as a solver-parameter and called OMEGA_TESTNUM.

The calculation process is considered divergent when Δ_{\max} equals or exceeds the comparative mean value. However, it would be impractical to set the relaxation factor back automatically every time this condition is satisfied before a certain minimum number of calculation steps has been performed. This quantity is a further criterium which must be met before an ω set-back occurs. The standard minimum number of iterations here is 23, defined as OMEGA_VETO.

An equation is ultimately considered solved when the deviation, Δ_{\max} , remains smaller than a defined limit for a continuous series of a prescribed number of iterations. This quantity is defined as TERM_NUM.

Finally, in order to "smooth" the results of calculation, a post-run of iterations is performed with a constant relaxation factor, $\omega = 1$ (defined as OMEGA_POSTRUN=1.0). An increase in this parameter should be avoided so as not to jeopardise the smoothing effect of post-calculation. The number of iterations of this stage is prescribed as POSTRUN=15.

results precision

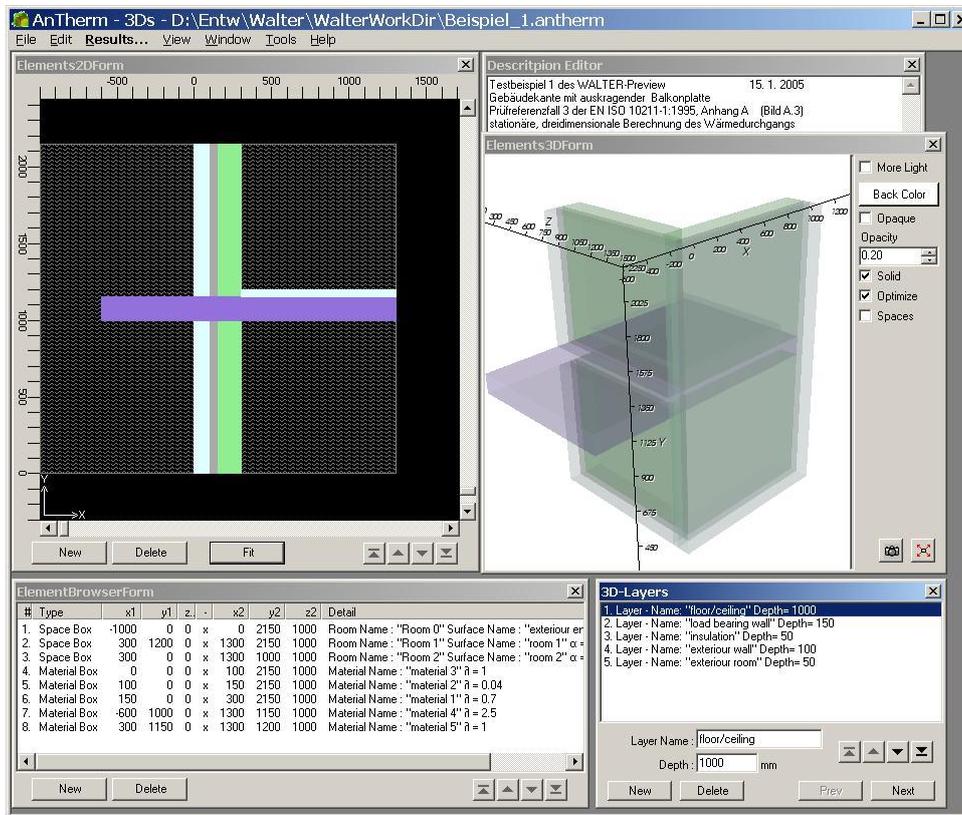
In summary, three primary factors control the precision of calculation results:

- [fineness of the grid](#) (calculation geometry).
- [stringency of the termination condition](#) (calculation time).
- precision of computation ("double precision").

The first two of these factors can be manipulated by the user, though the standard parameters used by AnTherm are adequate for the evaluation of most models to be considered.

Main application window

The main window provides the primary working area for all control elements of the application. While working the application the control is performed by using the [main menu](#) and various [editing, dialog](#) or [evaluation windows](#).
File->Neu.



On each application use its windows will be shown at their latest positions. On the absolutely first start of the application its windows will be shown at some tentative positions on the screen - as defined by a [pre-installed template](#). Windows can be freely moved and sized. This allows adjusting the layout according to user's needs. During application shutdown current windows positions are saved along with other [application settings](#).

The main window's title bar displays the name of the currently open project file - if the [file name](#) has been assigned already - together with the indication of project's type (2D, 3DS, 3D).

Depending on the [project type](#) of a [new, opened](#), imported or [converted](#) project the application will automatically expose [primary editing windows](#) suitable for the project type and/or it will dispose (close) all windows which are not useful for that one project type and processing stage.

Drag & Drop of a file onto the main window

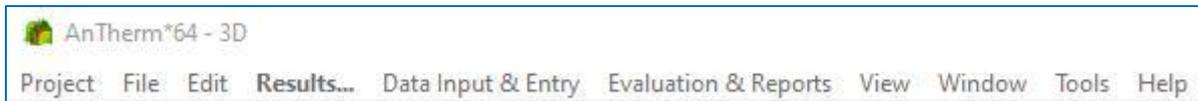
When a valid [AnTherm project file](#) is dropped onto the surface of the main window (after being dragged out from the windows file explorer) that file will be opened (equivalent to main Menu [File->Open...](#)). The name of the file will not be added to the [Most Recently Used List](#) unless the Shift- or Control-key is held depressed during the drop operation.

Remark: Dropping of a [Waebru component file](#) is also supported.

See also: [Main menu](#), [Editing and input windows](#), [Evaluation windows](#), [Dialog windows](#), [Miscellaneous control elements](#)

Main menu

Primary application functions are controlled and initiated from the main menu. The main menu is shown along the top edge of the [main window](#).

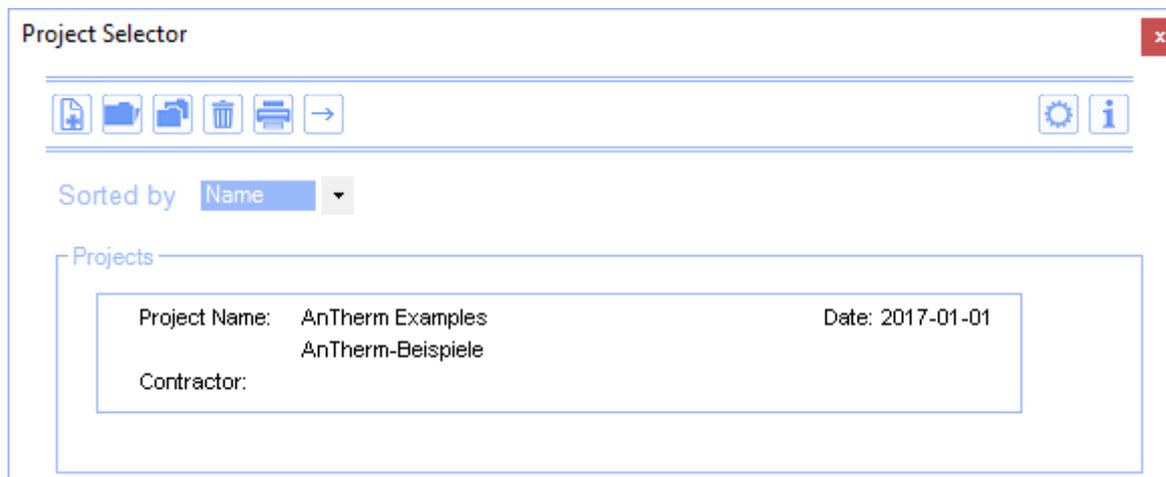


Where appropriate the main menu will be extended with further functions provided by an active window. For example the menu item *File* will contain the function *Print* if some [report window](#) is currently active. Likewise, when there is an active [report window](#) menu item *File*→*Export* will be extended with "*Export*→*Report*..." and while there are active 3D windows ([Elements3D](#), [Results3D](#)) the menu *File* will be extended with *3D-Navigation menu items*, and accordingly, the menu *File*→*Export* with *Export*→*Image 3D...* and *Export*→*Scene 3D...* with menu *Edit* extended by *Copy Image 3D*.

During some processing steps are executing (e.g. longer operations like calculating the solution or preparing evaluation data) functions of the main menu will be turned off. This will be made visible by greying out the menu characters. While the menu is deactivated it is not possible to use it.

Menu Project

From AnTherm Version 9 on, there is the Menu Project. This menu allows the user to select project files from an existing project (such as "AnTherm-Examples") and open them. Moreover, the user can create new projects and add project files to them. In addition, there are functions such as copy or delete. This menu is supposed to make it easier for users to preserve an overview of all projects and project files. But the traditional way of accessing project files directly via the Menu File is still enabled.



	New Project
	Open Project
	Copy
	Delete
	Print
	Forward (works only if you have already opened a project)
	Settings
	About

Once you open a project from the Project Selector Window (either by double-clicking the project button or clicking the button once and then clicking the "Open" button), the Project Details Window pops up:

Project Details
x

+ (Add Existing File)
📄 (New File)
📁 (Open Reports Folder)
🗑️ (Delete)
🖨️ (Print)
💾 (Save)
← (Back)

⚙️ (Settings)
ℹ️ (Info)

Project Detail

Project Name	AnTherm Examples
	AnTherm-Beispiele
Creation Date	2017-01-01

Contractor

Name	
Address	
Phone	

Files

Sorted by Name ▼

C:\Projekte\antherm v9_135_0 in entwicklung\AnTherm\x86\Release\Beispie... Date: 2014-03-13

- | | |
|---|---------------------|
| + | Add Existing File |
| 📄 | New File |
| 📁 | Open Reports Folder |
| 💾 | Save |
| ← | Back |

Menu File

Program functions put collectively into the *menu File* are used for file control. Occasionally there will be further menu items dynamically inserted to this menu, related to specific [editing windows](#) or [result/evaluation windows](#) (if such windows provide additional menu functions).



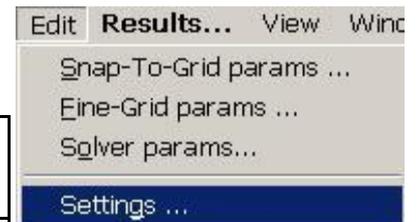
New→	Creates a new project - see also project types . Important: <u>prior to creating a new project it is recommended to save the current project's data!</u>
New→2D Project	Create a new project for 2D dimensional input only (input of the third dimension will be suppressed) Remark: A 2D project can also be thought as one layer of a " layered 3D project " with the thickness of 1000mm. See also Project types .
New→ Layered 3D Project	Create a new project built from series of 2D layers. The input of the third dimension is provided by entering the thickness of each layer and ordering layers in the layers list . See also Project types .
New→3D Project	Create a new 3D project. Component's construction will be composed of series of cubes placed in the 3D space. See also Project types .
Load...	Loading an existent project file
Load Recent→...	A project file to open can be chosen from the list of recently edited project files. <u>Remark:</u> The number of file names shown (and remembered by the program) can be changed in application settings . <u>Remark:</u> The list of recently used project files will be saved to the application settings file between program executions. <u>Remark:</u> If the project file listed does not exist the message will be shown and the cleanup of the list offered.
Save...	A project file to open can be chosen from the list of recently edited project files. <u>Remark:</u> The number of file names shown (and remembered by the program) can be changed in application settings . <u>Remark:</u> The list of recently used project files will be saved to the application settings file between program executions. <u>Remark:</u> If the project file listed does not exist the message will be shown and the cleanup of the list offered.
Save as...	Will ask you to provide a file name before project data is saved.
Import→	Files created with other applications can be imported into the application and the data will be transformed to a valid new project.
Import→Waebru.?BT...	Initiates an import from a WAEBRU component file (i.e. .2BT or .3BT file).
Import→aCad DXF...	Initiates an import from specially prepared DXF file .
Import→Heat2 DAT...	Initiates an import from a Heat2 text input file .
Import→Heat3 DAT...	Initiates an import from a Heat3 text input file .
Import→Kobru 86DAT...	Initiates an import from a Heat3 text input file .
Import→Image Underlay...	Initiates an import and registration of an image underlay/overlay . Remark: This will register an image for XY-Plane only (typical for 2D - and 3D-Layered Projects). To register three underlay images for the 3 planes of a 3D-Project (XY, YZ, ZX) use the context menu of respective quadrant within the Elements23 window .
Export→	Project data (also including result reports for example) can be "exported" to various file formats for further processing in other applications.
Export→Coupling Coeff. Matrix...	Initiates an export of the matrix of coupling coefficients (Leitwertmatrix) to a CSV file which can be then otherwise processed. Remark: This menu function is only available if there are results

	(the solution) already created and available.
Export→Report...	Initiates an export of a result report to some external file which can then be further processed. Remark: This menu function is only available if some report window is active.
Export→Image 2D... Export→Image 3D...	Allows saving of images to a picture file (PNG, TIF, JPG, BMP, GIF) and places the image onto the clipboard . Remark: This menu function is only available if some graphical window is active (e.g. Elements3D , Results3D , Elements2D , Elements23)
Export→Scene 3D...	Allows saving of a 3D scene to a 3D file (VRML, IV, OOGL, ...). Remark: This menu function is only available if there is some 3D window (Elements3D , Results3D) active.
Convert→	Project data can be " converted " to some other project type .
Convert→Layered 3D Project	Initiates the conversion of currently edited project data to a new layered 3D project . Remark: Not each project can be converted to that project type. See also Project types , conversion variants
Convert→3D Project	Initiates the conversion of currently edited project data to a new 3D project (in a 3D project Z coordinates of elements can be entered directly). See also Project types , conversion variants
Convert→2D Project	Initiates the conversion of currently edited project data to a new 2D project . Remark: Only projects homogenous in the third dimension (Z axis direction) can be converted to a 2D project (e.g. layered 3D project containing only one layer, or a 3D project of which all elements share same Z coordinates). See also Project types , conversion variants
Convert→Revolve to 3D...	Initiates the conversion by revolution (rotation) of currently edited project data of a 2D project to 3D . See also Project types , conversion variants
Print Report...	Initiates printing of some report . Remark: This menu function is only available if some report window is active.
Exit	Initiates the shutdown of the application. Warning: please save project data prior to exiting the application!

Menu Edit

Program functions placed collectively in the *menu Edit* are used for adjusting various processing parameters of the application.

Fine-Grid params...	Shows the dialog used to adjust fine grid generation settings of the current project.
Solver params...	Shows the dialog used to adjust settings of the solver engine in current project.
Settings...	Shows the application settings' dialog .



Occasionally there will be further menu items dynamically inserted to this menu, related to specific [editing windows](#) or [result/evaluation windows](#) (if such windows provide additional menu functions).

Menu Results... (or the F5 function key)

The *menu Results...* initiates calculation of results for the currently processed project. Occasionally you will be asked to save the project data to a [project file](#) and to set parameters for [fine grid](#) and for the [solver engine](#).

Before the calculation is actually started the application executes series of [validation checks](#) on the model construction entered.

The calculation process can be observed in the [Solver window](#). Which calculations are possible depends on actual model data and valid [application license](#) features.

After the calculation has successfully completed the application will show the [coupling coefficients report \(Leitwerte\)](#) and will request the input of [boundary conditions](#). Following the later further [results reports and evaluations](#) can be performed.

See also: [Solver window](#)

Menu → Data Input & Entry

The submenu *Data Input & Entry* groups functions used to open various [editing and input windows](#) of the application. In the normal case most of this windows can be opened in the processing context from other windows directly too. The menu *Data Input & Entry* provides in that sense an additional function only.

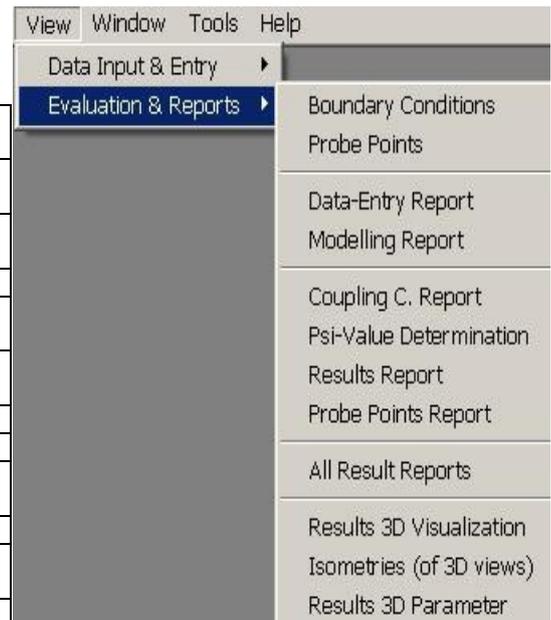
Primary Edit Windows	Opens primary input windows essential for the current Ptype of project (Description , Element selection , Layers , Elements 2D , Elements 3D , etc.) Remark: Which windows are considered essential can be set by user in the application settings for each different project type selectively.
All Edit Windows	Opens all input and edit windows of the application
Description	Opens the window " Description "
Edit Element	Opens the window " Element Editor "
Elements Browser	Opens the window " Elements browser "
Elements 2D	Opens the window " Elements 2D "
Elements 3D	Opens the window " Elements 3D "
Elements2/3D	Opens the window " Elements 2/3D "
Materials	Opens the window " Materials "
Surfaces	Opens the window " Surfaces "
Layers Browser	Opens the window " Layers Browser " <i>Remark: Not offered for 3D project</i>
Translate Elements	Opens the window " Translate "
Stretch Elements	Opens the window " Stretch "
Rotate Elements	Opens the window " Rotate "
Mirror Elements	Opens the window " Mirror "
Status Report	Opens the window showing the " Status report "
Material Database	Opens the window " Baustoffstammdaten "
Isometries (of 3D views)	The list of isometries is shown in the window Isometries .



Menu → Evaluation & Reports

The submenu *View*→*Evaluation & Reports* groups together functions used to open various results- and [evaluation windows](#) of the application.

Report builder	Opens the window "Report builder"
Boundary Conditions	Opens the window " Boundary Conditions "
Probe Points / Min/Max	Opens the window " Probe Points /Extreme (min/max) Locations "
Data-Entry Report	Opens the window " Data-Entry Report "
Modelling Report	Opens the window " Modelling Report "
Results Report	Opens the window "Results Report"
Coupling Coeff. Report	Coupling Coeff. Report Opens the window "Thermal Coupling Coefficients" Report
Surface Temperatures	Opens the window "Surface Temperatures"
Results3D Visualisation	Opens the window " Results 3D "
Isometries (of 3D views)	The list of isometries is shown in the window Isometries .
Results 3D Parameter	The choice list of evaluation parameters is shown in the window Results3D parameter



Menu View→2D Navigation

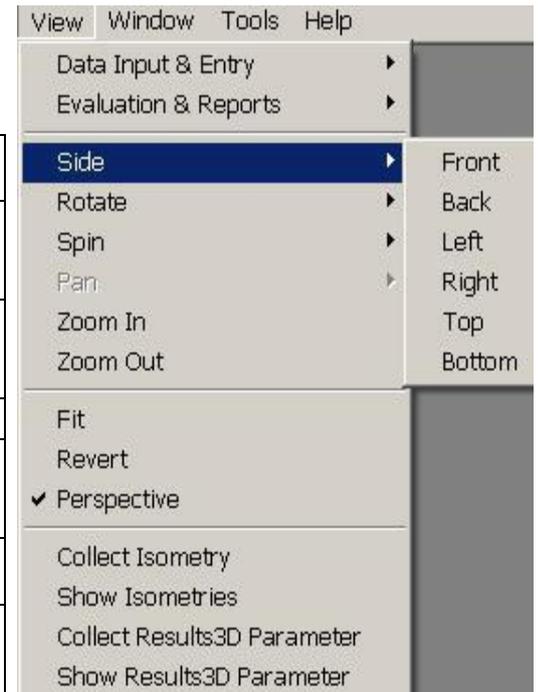
The Menu → View is only for an active window. Functions of this menu control the movement of the element display within the view.

Zoom In Zoom Out	The view can be „scaled“.
Fit to window	Adjust the position and scaling in such way, that the whole construction is visible within the window.

Menu View→3D Navigation

Menu items of 3D navigation are displayed as sub-items of the menu *View* if there is a 3D window active ([Elements3D](#), [Elements23](#), [Results3D](#)). Functions of this menu control the movement of the camera (observer) within the three dimensional scene.

Side	Turns the 3D view. The camera is moved to one of <i>side view positions</i> (Front, Bottom, Top etc.)
Rotate	The camera is moved on the surface of a sphere (the distance to the object's centre does not change). Turns are executed in 30° steps
Spin	The camera is moved on the surface of a sphere (the distance to the object's centre does not change). Turns are executed in 30° steps
Pan	The camera pans.
Zoom In/Out	The camera moves forwards or backwards. With the parallel projection it is equivalent to scaling the view.
Fit	The scaling and position of the view is adjusted to fit the whole building construction within the window's view.
Revert	Restore the original standard camera position. X/Y plane parallel to view plane. Z axis point towards the viewer. Y axis points upwards. Scaling adjusted to fit the view.
Perspective	Swiches the 3D display between <i>perspective projection</i> and <i>parallel projection</i> . The parallel projection is useful for 2D components or slice views (see also "Side Views" here above).
Collect Isometry	The current position of the 3D display (in Elements3D , Elements23 or Results3D) is added to the project specific list of isometries . The list of isometries can be edited in the window Isometries (each entry can be named for example) or one of listed isometries can be selected from the list and applied to 3D views.
Show Isometries	The list of isometries is shown in the window Isometries



See also: [3-D Navigation \(control panel\)](#), [3D Navigation in 3D windows](#), [Track-Ball Rotation](#), [Results3D window](#), [Elements3D window](#), [Elements23 window](#), [Isometries window](#)

Menu View→Results 3D

Menu items of Results 3D are displayed as sub-items of the menu View (see above) if there is Results3D window_active. Functions of this menu control settings (parameters) of the evaluation.

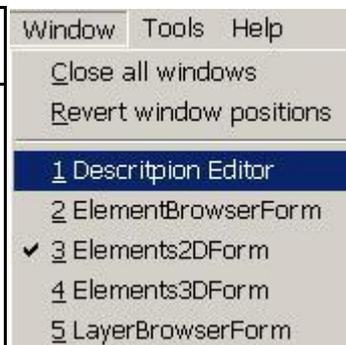
Collect Isometry	The current position of the 3D display is added to the project specific list of isometries . The list of isometries can be edited in the window Isometries (each entry can be named for example) or one of listed isometries can be selected from the list and applied to 3D views.
Show Isometries	The list of isometries is shown in the window Isometries .
Collect Results3D Parameter	Current settings (parameter) of a graphical evaluation in the Results3D window are saved and added to the project specific choice list of evaluation parameters .
Show Results3D Parameters	The choice list of evaluation parameters is shown in the window Results3D Parameter .

See also: [Results3D Parameter window](#), [Isometries window](#), [Results3D window](#), [General \(control panel\)](#), [3D Navigation \(control panel\)](#)

Menu Window

The *Window menu* holds functions used for management of windows. In addition a list of open windows is pinned at the bottom of the menu and thus allows direct access to this windows (e.g. activation and bringing the window to front).

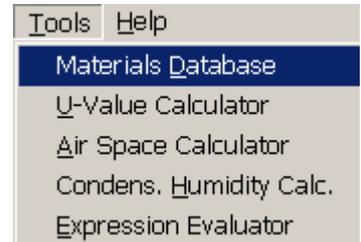
Close all windows	Closes all application window (except from main application window).
Revert window positions	Window positions will be reset to its initial positions as of the very first application start-up. <u>Remark</u> : You must additionally confirm that request before it is executed. See also: Initial template of window positions AnTherm.exe.LayoutPersistence.xml
List of window names	List of all open windows. It is used to activate any of already open windows. <u>Remark</u> : The list is shown only in the MDI application mode .



Menu Tools

The *menu Tools* collect various application *tools*. This additional function collectively shown in this menu have no direct impact on edited construction and are used as help for calculations.

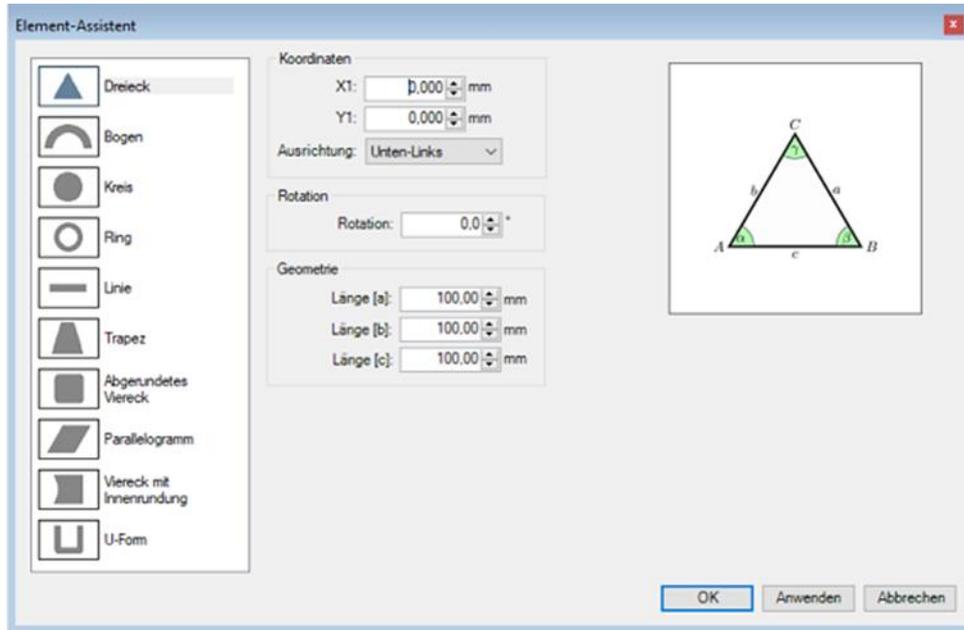
Materials Database	Opens the window " Materials Database "
Materials Database(Custom)	Opens the window " Materials Database " Custom
Psi-Value Calculator (manual).	The manual tool to calculate a Psi-Value used for rapid calculation of the linear thermal transmittance of arbitrary thermal heat bridge.
Psi-Value Calculator (automatic)	The automatic tool to calculate a Psi-Value used for rapid calculation of the linear thermal transmittance of arbitrary thermal heat bridge.
Chi-Value Calculator	The manual tool to calculate a Chi-Value
U-Value Calculator	A tiny tool to calculate a U-Value of a layered construction .
Air Cavity Calculator	A tiny tool to quickly calculate equivalent heat transfer coefficient of small air spaces and cavities based on dimensions of the cavity and heat stream direction.
Condensing Humidity Calculator	A tiny tool to calculate condensing humidity (dew point) based on input of surface temperature and space (air) temperature.
Periodic / Harmonic Data Editor	Provides the means of creating, editing and managing periodic data required as boundary conditions to transient evaluation (requires TRANSIENT license feature)
Inhomogenous Calculator	The Inhomogenous Layer Calculator can be used to compute the features of a thermally equivalent homogenous layer. Possible applications include brick masonry and wooden walls.
Expression Evaluator	A tiny pocket calculator able to calculate simple expression (formulas) like 1/25 or Math.Sin(3.14).
Solution Folder Cleaner	A tool used to cleanup unneeded simulation folders .
New AnTherm Instance	The second (or next) new instance of AnTherm will start in parallel to the current. Remark: Neither current project data nor application settings are saved! The new instance loads with earlier settings. Remark: This function is available only if the setting "Multiple app-instances allowed" has been activated within application settings .
New Instance Saved Project	After requesting the user to save project data (equivalent to File->Save as...) the second (or next) new instance of AnTherm will start and load the project data just saved. Remark: Current application settings are not saved! The new instance loads with earlier settings. Remark: This function is available only if the setting "Multiple app-instances allowed" has been activated within application settings .
Settings...	Shows the application settings' dialog.



Menu Element-Wizard

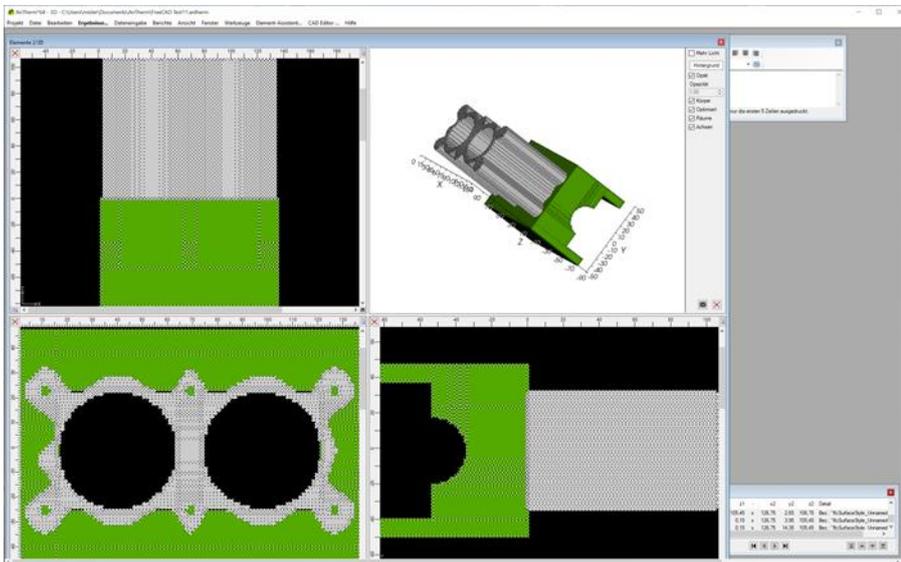
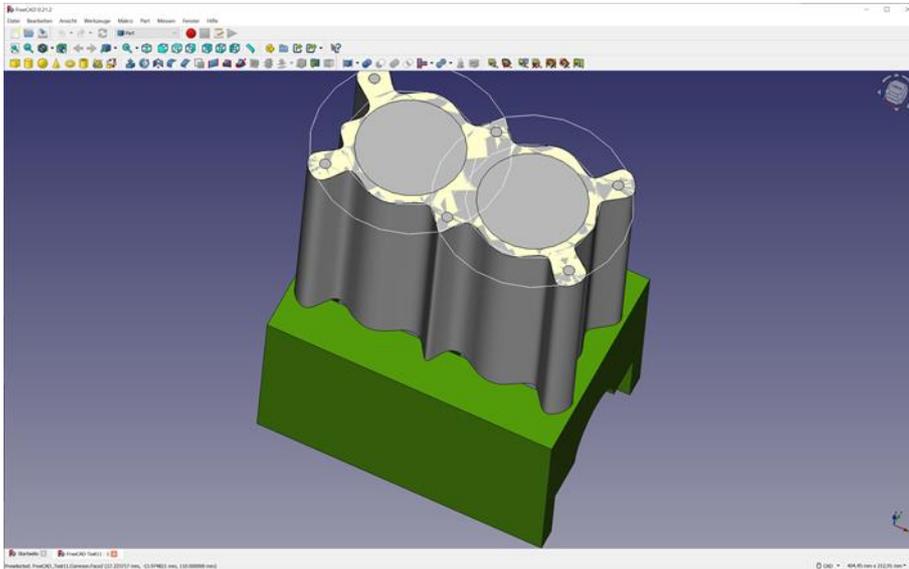
With the help of "Element Wizard" you could model geometric figures (triangle, arc, circle ring) in AnTherm.

If you want to rotate the new elements, please do this directly in the Element Wizard, then the rotation will work quickly, otherwise it will take longer.



Menu CAD Editor

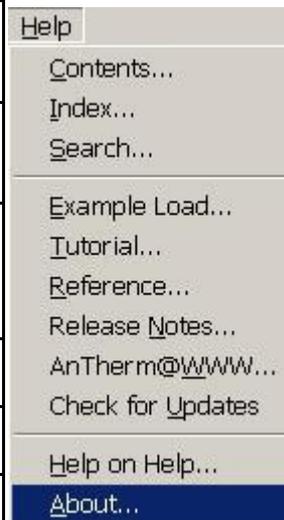
CAD editor The CAD editor is used to model various projects or to import from other file formats. In order to transfer the model from the CAD editor to AnTherm, it simply needs to be saved, either explicitly using the "Save" menu item or when exiting the CAD editor by confirming the save dialog with Yes.



Menu Help

In the *menu Help* you will find links to various *help* and *information* functions of the application

Contents...	Shows the help window and the table of contents of the application's help system.
Search...	Shows the help window and a keyword search window of the application's help system.
Example Load...	Using the same function as of File->Load a list of files in the examples folder (created during installation) will be offered. This allows quick access to example files deployed together with the application installation.
Tutorial...	Shows first page of tutorials (cook book) .
Reference....	Shows first page of short reference .
Tip of the Day...	Shows the dialog window "Top of the Day" .
Release Notes....	Shows first page of the change log (history of changes to applications).
AnTherm@WWW...	Shows the AnTherm's web page on internet (requires internet connection). Currently the page at the URL http://www.antherm.kornicki.de/ is shown.
Check for Updates...	Checks if there are updates or new version of the application (requires internet connection). Currently the page at the URL http://www.antherm.kornicki.de/support/CheckForUpdates/ is shown..
Help on Help...	Shows the help window describing the usage of the help system.
Info-„About“...	Shows the About window of the application (application adn system environment, license, etc.).



Remark: With the [application setting "Help Window shown TopMost"](#) the user can decide if the Help window will be shown topmost to all application windows and automatically minimize with the application (which is the windows default behaviour) or it can be set behind the application window via typical desktop window manager (default application setting).

See also: [The main application window](#), [Editing windows \(Input windows\)](#), [Evaluation windows](#), [Dialog windows](#)

Editing windows (Input windows)

Editing (or Input) windows are used for entry, modification or adjustment of data of one modelled constructions.

Following editing windows are used in most cases:

- [Description](#)
- [Element selection](#)
- [Elements 2D](#)
- [Elements 3D](#)
- [Elements 2/3D](#)
- [Element editor](#)
- [3D-Layers](#)

and most of them share the common [Context Menu of Element Editing Functions](#).

During data entry further windows might be helpful:

- [Boundary conditions](#)
- [Probe points](#)
- [Materials](#) and [Material](#)
- [Surfaces](#) and [Surface](#)
- [Translate](#) together with [Rotate](#), [Mirror](#), and [Stretch](#) (i.e. [transforms](#))
- [Materials Database](#)

together with following tools:

- [Ψ-Value \(Psi Value\) Calculator \(tool\)](#)
- [U-Value calculator \(tool\)](#)
- [Air cavity calculator \(tool\)](#)
- [Condensing Humidity \(Dew point\) calculator \(tool\)](#)
- [Periodic / Harmonic Data Editor](#)
- [Expression calculator \(tool\)](#)

In addition to the above editing/input windows there are [further dialog windows](#) used for more input.

Undo (Ctrl-Z) and Redo (Ctrl-Y) of editing actions



Editing windows provide the [Undo action](#) via the menu entry or a **keyboard shortcut Ctrl-Z**.

The editing sequence up to 1000 editing actions can be easily undone to revert to the input state valid prior to the respective action.

Actions Undone recently can be redone in the same order ([Redo, Ctrl-Y](#)).

Executing an Undo or Redo action will expose the editing window the action originated from or primarily responsible for the action.

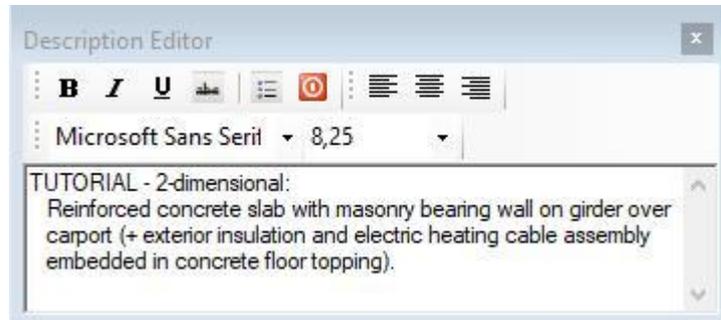
 Undo (Ctrl-Z)	Undoes (reverts) the latest action(s) in the given context. <u>Remark:</u> Undo operations are available to element and material property editing actions.
 Redo (Ctrl-Y)	Redoes (reapplies) the action lately undone in the given context.

Remark: The undo buffer ist kept within the context of the current project and will be emptied upon creation, loading, converting etc. actions applied to the project itself.

See also: [Dialog windows](#), [Evaluation windows \(Results windows\)](#), [The main window](#), [Context menu of element editing](#),

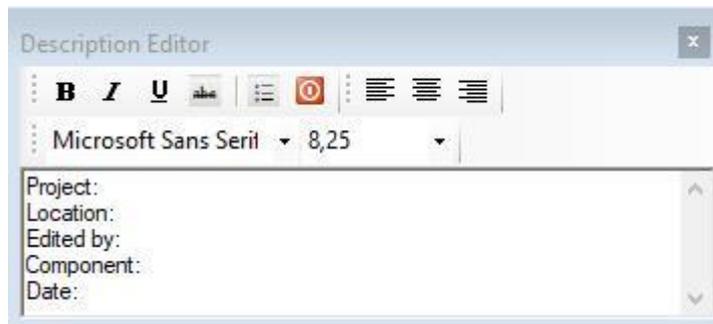
Description - window

In the window description you input a short description of the project - as one or multiple lines of text.



The project description is output in the heading of all [project reports](#).

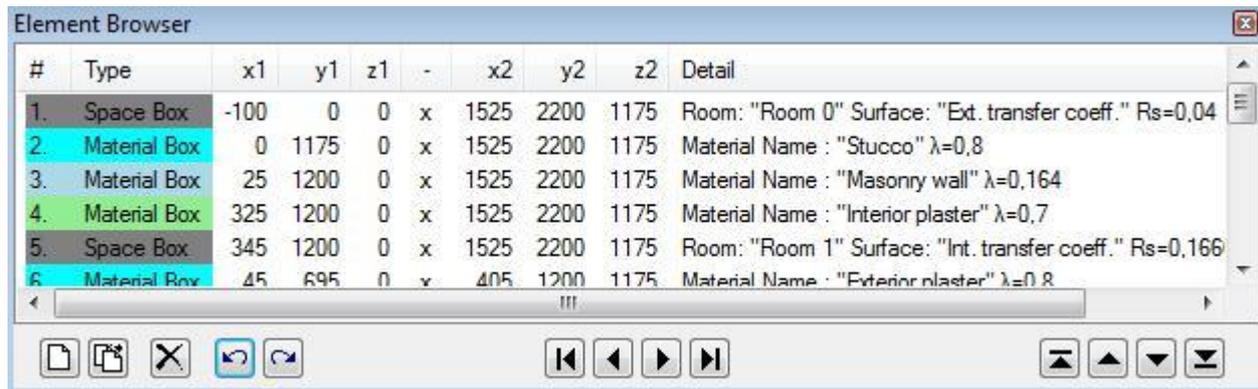
Project description can be arbitrarily changed in this window.



See also: [Reports](#), [Standard project template file projekt.xml](#)

Element selection (browser) - window

Element selection (element browser) window shows the list of all input elements of the construction (or elements of the [active layer](#)) in specific order of [overlapping](#).



From that window you can select elements for editing, insert new elements, reorder elements in their overlapping order (promote or relegate) and also delete elements.

	Element list	List of all elements entered (or elements of the active layer) in specific order of overlapping.
	New	Inserts a new element (of type "Empty") to the end of the list.
	Duplicate	Inserts a copy of currently selected element(s) into the element list. Further details on duplicate function are provided in the description of the context menu of element editing functions .
	Delete	Deletes selected element(s) from the list. After deleting one selected element the selection move to the one before the deleted. When first element has been deleted the one becoming first after deletion is selected. If more then one element are selected actual deletion is executed if additionally confirmed by the user. See also: Cut elements to clipboard
	Undo(Ctrl-Z)	Undoes (reverts) the latest action(s) in the given editing context .
	Redo(Ctrl-Y)	Redoes (reapplies) the action lately undone in the given editing context .
	First element	Selects the first element in the element list.
	Previous	Selects an element which is previous to the currently selected in the element list.
	Next	Selects an element which is next to the currently selected in the element list.
	Last element	Selects the last element in the element list.
	Relegate to Backmost	Reorders selected element(s) in the list to the "backmost" position. The element is then located at the beginning of the list thus now being potentially overlapped by all other elements. If more elements are selected then each one is relegated by the same distance as the backmost selected one. This action is inhibited if the chosen element does already occupy the first (backmost) location in the list.
	Relegate	Reorders selected element(s) in the list one position "backwards". The element is then located earlier in the list thus now being potentially overlapped by another element which have thus been promoted to the location of the element relegated. If more elements are selected then each one is relegated by one position in the list order. This action is inhibited if the chosen element does already occupy the first (backmost) location in the list.
	Promote	Reorders selected element(s) in the list one position "towards front". The element is then located later in the list thus now potentially overlapping another element which have thus been relegated to the location of the element promoted. If more elements are selected then each one is promoted by one position in the list order. This action is inhibited if the chosen element does already occupy the last (front most) location in the list.

 Promote to Frontmost	Reorders selected element(s) in the list to the "frontmost" position. The element is then located at the end of the list thus now potentially overlapping all other elements. If more elements are selected then each one is promoted by the same distance as the frontmost selected one. This action is inhibited if the chosen element does already occupy the last (front most) location in the list.
Click right mouse button	Exposes the context menu of element editing functions
Click left mouse button	Selection of an element in the element list.
Double-click left mouse button	Selection of an element in the element list and exposing the Element editor window for that selected element.
Click CTRL+left mouse button	Extend the selection with the one element clicked or removes the element clicked from the current selection.
Click SHIFT+left mouse button	Extends the selection with all elements between the one clicked and the one selected earlier.
Drag left mouse button	Selecting of one or more elements touched with the mouse pointer during the drag operation.

Elements list

Each line of the elements list represents one input element. The line shows the **type of the element** (Space, Material, Power source, Empty), its position given by entered coordinates (x1,y1,z1)-(x2,y2,z2), name of the material used or of the surface, space or power source together with its physical properties such as heat conductivity of the material or surface heat transfer coefficient.

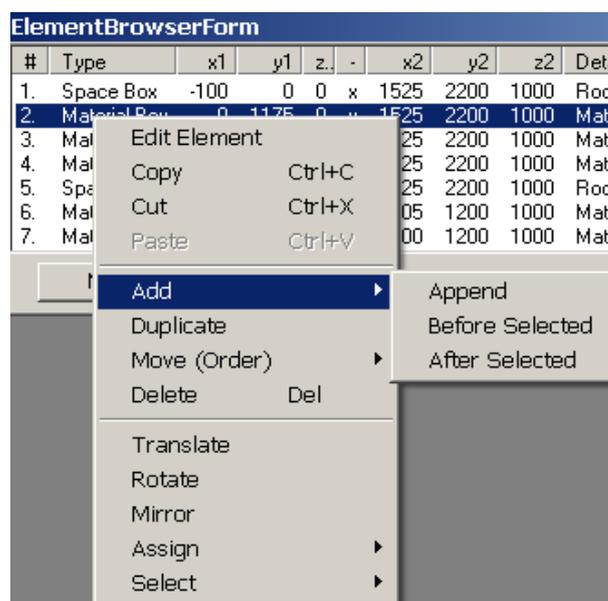
When the mouse pointer hovers over one element the properties of the element under the pointer are shown in a tooltip (small informational popup window). The tooltip text disappears after 5 seconds or on any mouse movement or click.

Columns Number# and Type are rendered by approximating the colour of an element (the text is rendered in the contrast colour depending on the value of the element colour).

Element editing

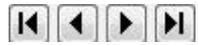
A double click over a element's line in the elements list will open the **Element editor window** or expose it if it is already open.

Clicking the right mouse button over the elements list will reveal (possibly very complex) **context menu of element editing** allowing choice of element processing functions. This context menu and its commands are common to all **element editing windows** and **is described separately**.



Element selection

Clicking with left mouse button with the mouse pointer placed over a line within the elements list will select the element clicked Keyboard's arrow keys can also be used for such single element selection.



Element selection buttons can be used to select single elements: the first in the order, previous to the currently selected, next to the current or the last one in the element list.

Selection of multiple elements

While holding the control-key (CTRL) one can click on one line of the element list to add that element to the selection or to remove it from the current selection.

While holding the shift-key (SHIFT) one can click on one line of the element list to select all elements between the line clicked and the element selected just beforehand.

The selection can also be extended by dragging with the pointer over elements while holding the left mouse button pressed.

Reordering (promoting/demoting) of elements

The order of elements shown in the element list is significant for resulting overlapping effect of elements intersecting one another. Changing the order of elements in the element list leads to change in the overlapping effect. By overlapping elements in the order of input one receives automatically dynamic intersection of elements positioned earlier in the order.



Command buttons "Promote" or "Relegate" are used to move selected elements in the list and thus they change the order of overlapping intersections.

Promoting and relegating is also possible if several elements are selected. Each selected element is shifted by one position until the block of selected elements reaches the start or the end of the list. If the first or last selected element has already reached beginning (or end) of element list no further reordering action in this direction can be performed any more - this limitation provides the ability to revert the operation in the opposite direction to eventually restore to the original situation (order).

See also: [Element Editor](#), [Order of overlapping](#), [context menu of element editing](#), [Types of elements](#), [Layers window](#)

Elements 2D - window

The *Elements 2D window*, together with [element list](#) and the [element editor](#) provide the central front end of the application used for data input of a modelled building construction component. Pressing the **right mouse button** exposes the [context menu](#) offering all [element editing functions](#).

The *Element 2D window* shows graphically the list of all input elements (or elements of the [active 3D-layer](#)) in the specific order of overlapping.

Remark: This window has its primary use for [2D- and 3D-layered models](#) only. For pure [3D models](#) it will only show the Z-view onto the model - thus it is advised to use the [Elements 23 window](#) to process pure 3D models.

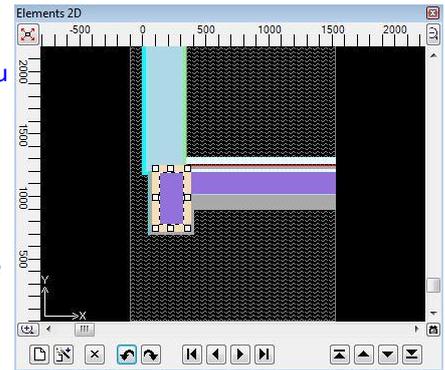
On one hand the window provides an instant graphical view onto the component under construction. On the other it is used as an input device together

with the mouse as a tool to edit current construction. Graphical sizing of elements is supported with the [snapping grid](#).

From this window one can select elements for processing, add new elements, change the overlapping order (promote or relegate) of elements or delete them.

The [2D view can be copied onto the clipboard](#) to be *pasted* later into another application (e.g. Word) from *the clipboard* - for example for the purpose of integrating this view into some user's report. The image can be [saved to a picture file](#) also.

Remark: Functions offered in this window are by far comparable to those offered by [Element selection \(browser\) window](#), because both windows show *the same list of elements*, but in two different representations - graphical and textual.



Element graph	Graphical representation of all elements input (or elements of the active layer) in the specific order of overlapping.
New	Inserts a new element (of type " Empty ") to the end of the element list .
Duplicate	Inserts a copy of currently selected element(s) into the element list . Further details on duplicate function are provided in the description of the context menu of element editing functions .
Delete	Deletes selected element(s) from the list. After deleting one selected element the selection move to the one before the deleted. When first element has been deleted the one becoming first after deletion is selected. If more then one element are selected actual deletion is executed if additionally confirmed by the user. See also: Cut elements to clipboard
Undo(Ctrl-Z)	Undoes (reverts) the latest action(s) in the given editing context .
Redo(Ctrl-Y)	Redoes (reapplies) the action lately undone in the given editing context .
	Selects an element which is first, previous to the currently selected, next to it or last in the element list .
Relegate to Backmost	Reorders selected element(s) in the element list to the "backmost" position. The element is then located at the beginning of the list thus now being potentially overlapped by all other elements. If more elements are selected then each one is relegated by the same distance as the backmost selected one. This action is inhibited if the chosen element does already occupy the first (backmost) location in the list.
Relegate	Reorders selected element(s) in the element list one position "backwards". The element is then located earlier in the list thus now being potentially overlapped by another element which have thus been promoted to the location of the element relegated. If more elements are selected then each one is relegated by one position in the list order.

	This action is inhibited if the chosen element does already occupy the first (backmost) location in the list.
 Promote	Reorders selected element(s) in the element list one position "towards front". The element is then located later in the list thus now potentially overlapping another element which have thus been relegated to the location of the element promoted. If more elements are selected then each one is promoted by one position in the list order. This action is inhibited if the chosen element does already occupy the last (front most) location in the list.
 Promote to Frontmost	Reorders selected element(s) in the element list to the "frontmost" position. The element is then located at the end of the list thus now potentially overlapping all other elements. If more elements are selected then each one is promoted by the same distance as the frontmost selected one. This action is inhibited if the chosen element does already occupy the last (front most) location in the list.
Click right mouse button	Exposes the context menu of element editing functions
Click left mouse button	Selection of an element in the element list for further processing.
Turning the mouse wheel 	The view will be "scaled" - displayed area will be zoomed. Remark: The scaling operation focuses around the mouse pointer. This allows zooming of the area of interest directly followed by ability to pan the view with the mouse. Remark: If one of the scrollbars has focus, then turning the mouse wheel will result in scrolling (instead of zoom). Remark: During drag operations of the editing mark the function of scaling by the wheel mouse turns is suspended
 Fit	The position (scroll) and scaling (zoom) of the view will be adjusted to display the whole component in the window. Remark: If there is an underlay registered and visible its bound will be taken into account also.
Drag with left mouse button depressed (not on the edit marker)	The view will be „scrolled“ - displayed area is panned following the mouse. Remark: You can use scroll bars to pan the view also.
Drag with left mouse button depressed (on the edit marker)	Translates the element or changes the position of element's side or its corner. Remark: If snapping grid is active, translations are adjusted to valid snapping grid line positions also.
Double-click left mouse button	Selection of an element in the element list and exposing the Element editor window for that selected element.
Click CTRL+left mouse button	Extend the selection with the one element clicked or removes the element clicked from the current selection
Drag SHIFT+left mouse button	Selection of element(s) with a lasso
Drag SHIFT+CTRL+left mouse button	Extending the selection of element(s) with a lasso
CTRL+C, CTRL+X, CTRL+V	Copy or Cut selected elements to the clipboard or Paste elements from the clipboard.
 Copy Image	Allows saving of the image to a picture file (PNG, TIF, JPG, BMP, GIF) and places the image onto the clipboard. See also: application setting "clipboard image Save-To-File option"

The graphical display

The window displays a 2-dimensional slice through the building component. Elements are drawn in the order of their appearance in the [element list](#). This results possibly (and as desired) in the effect of overlapping of elements being immediately visible on the screen

When the mouse pointer hovers over one element the properties of the element under the pointer are shown in a tooltip (small informational popup window). The information shown applies to the visible (i.e. not overlapped) element. The tooltip text disappears after 5 seconds or on any mouse movement or click.

Each element is displayed differently depending on its [element type](#) and defined colouring:

- material elements are shown filled in their [defined colour](#)
- empty elements are shown as edge line surrounding the „empty area“ and with two diagonal lines
- space elements are filled with waved lines
- power source elements are filled with a fine net of lines and coloured in red.



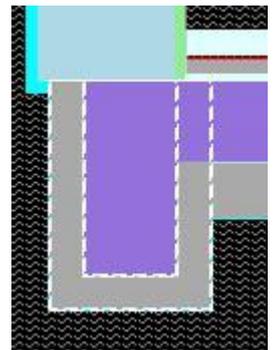
Due to the [overlapping effect](#) (which is also desired) one or more elements might be fully obscured and thus not visible at all. Such (fully obscured) elements are emphasized with a selection marker only if they are [selected](#) in the element list.

Selected elements are emphasized with selection markers. When only one element is selected a [editing marker](#) is shown instead and can be used to size or move that one element.

Remark: Line representation used to show elements with zero extent might be too thin at smaller view scaling and not visible. Such elements are visible only when selected.

The display can be adjusted according to users needs as required:

- Scrollbars allow panning of the view (which is also possibly by simply dragging the view with the mouse).
- Turning the mouse wheel will scale the view around the point below the mouse pointer (enlarged or shrunken).
- The fitting function adjusts the position and scaling in such way, that the whole construction is visible within the window.
- The colour of the background (which is by default set to black) can be changed within the [application setting](#) "[Background 2D-colour](#)".
- A [registered image](#) can be displayed as an underlay or as partly transparent overlay for better orientation during creation of the model.



Element selection

One or more elements in the [element list](#) can be selected. Selection can also be performed within the graphic view in this window:

- by using the [lasso function](#) one can select many elements for further [processing](#)
- a mouse click while the mouse pointer rests over one element will select that element which then will be emphasized with an [editing marker](#).
- Remark: If there are more elements stacked below the mouse pointer, each next mouse click will select the one behind the one currently selected (i.e. in the opposite order to the overlapping sequence).
- a mouse click while the control key (CTRL) is depressed will add the element below the mouse pointer to the list of selected elements.
- by element selection buttons  to select single elements from the [element list](#).

Lasso

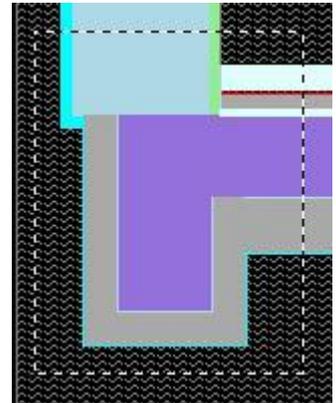
The lasso provides the function for [selecting](#) elements for further processing.

The lasso function will be activated when the shift-key (SHIFT) is held depressed together with left mouse button typically followed by dragging the mouse.

All elements fully inside of the lasso will be selected when the mouse button is returned to its resting position.

When the control key (CTRL) is also held depressed (SHIFT+CTRL+left mouse button) the selection will then be extended with elements matched with lasso.

To cancel the lasso selection process (i.e. to keep the last selection active) simply **free the SHIFT key** while the mouse button is still kept depressed.



Element processing

A double click with the mouse over an element will open the processing window designed for elements - the [element editor](#).

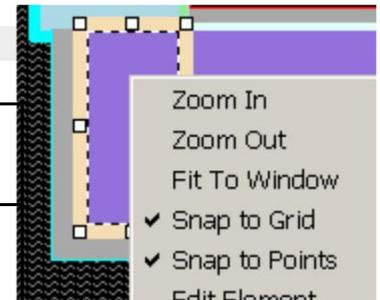
When the right mouse button is depressed in the graphical window, the [context menu of element processing](#) is exposed offering extensive list of element processing functions for choice.

Elements can be [selected](#) with a single mouse click or with the [lasso function](#).

If there is only one element selected its position can be changed by dragging the [editing marker](#) with the mouse.

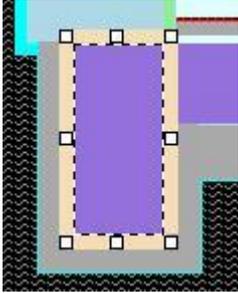
Functions of the context menu

Zoom In	The view can be „scaled“.
Zoom Out	This is an alternative to using the mouse wheel .
Fit to window	Adjust the position and scaling in such way, that the whole construction is visible within the window.
Snap to Grid	The function of the snapping grid will be turned on or off. The "check mark" shows the active setting of this function. See also: Snapping grid parameters
Snap to Points	This is a snapping function , which snaps to already existing coordinate points and can be turned on or off here. The "check mark" shows the active setting of this function. See also: Snapping grid parameters
Snap-To-Grid params...	Shows the dialog used to adjust current snapping grid settings .
Underlay/Overlay→	Functions respective to underlay/overlay. <u>Remark:</u> The planar underlay is registered at the plane shown within the window. Within the context of Elements23 window (i.e. a 3D-Project) up to three underlays can be registered within respective quadrants .
Underlay/Overlay→ Image Register	Offers the dialog "Register Underlay Image" to import an image file.
Underlay/Overlay→ Image Hide	If there is an underlay image registered and shown it will be hidden.
Underlay/Overlay→ Image Drop	If there is an underlay image registered its registration data will be removed from the current project.
Underlay/Overlay→ Image Underlay	If there is an underlay image registered it will be shown as underlay (being occluded by elements entered).
Underlay/Overlay→ Image Overlay 100%	If there is an underlay image registered it will be shown as overlay (occluding elements entered) either opaque (100%) or with 50% or 20% of transparency value. The currently selected transparency value is marked with a checkmark left to the menu item. See also: Dialog "Register Underlay Image" to import an image file
Image Overlay 50%	
Image Overlay 20%	



Many further functions of this [context menu](#) are common to all element processing windows and are [described separately](#).

Editing marker



The editing marker is used for adjusting the position of one element. It is displayed as rectangle emphasized with a thick coloured edges.

Dragging an edge with the mouse will translate (move) the complete edit marker (and thus the element will also move). If however edge centres or corners are dragged (these are emphasized with small rectangles) only the particular edge or corner will change its position.

By pressing the **ESC-key** the change initiated by dragging the mouse can be reset to its original state at any moment.

Remark: If [snapping to grid](#) is active, changes to the position are restricted to current snapping grid lines.

Remark: When mouse pointer hovers over edges lines of the editing marker the properties of the element edited are shown in a tool-tip. The information shown applies to the edited element. The tool-tip text disappears after 5 seconds or on any mouse movement or click.

A double-click over the edge of the marker will expose the [Element editor window](#).

Snapping grid

The [snapping grid](#), if active, restricts coordinate values which can be reached by moving the editing marker.

The snapping grid can be set up in such a way that only integral coordinates are allowed (each 1 mm) or only coordinates in integral multiplies of ten (each 10 mm) - etc. The [setting of snapping grid](#) can be adjusted from [main menu](#) via [Edit→Snap to Grid params...](#) . Activation and deactivation of snapping grid is also possible in the [context menu](#).

When the function of snapping grid is turned off movements of editing marker will result in coordinate values calculated by the inverse transform of pixel coordinates at the mouse position. This transformation can result in numbers with large number of decimal places which, in most cases, will not correspond to desired element coordinates.

See also: [Element Editor](#), [Element selection \(browser\) window](#), [Editing \(input\) windows](#), [Context menu of element processing](#), [Snapping grind parameters](#), [Coordinate system](#), [Export Image 2D](#), [Copy Image 2D](#)

Element Editor

The Element Editor window provides a central input function of construction input. Depending on the [type of input or edited element](#) different input fields are activated or shown.

Note: You shall switch between input fields by using the TAB-key. Using that key confirms the data entry. On the other hand it provides a good alternative compared to permanent mouse use.

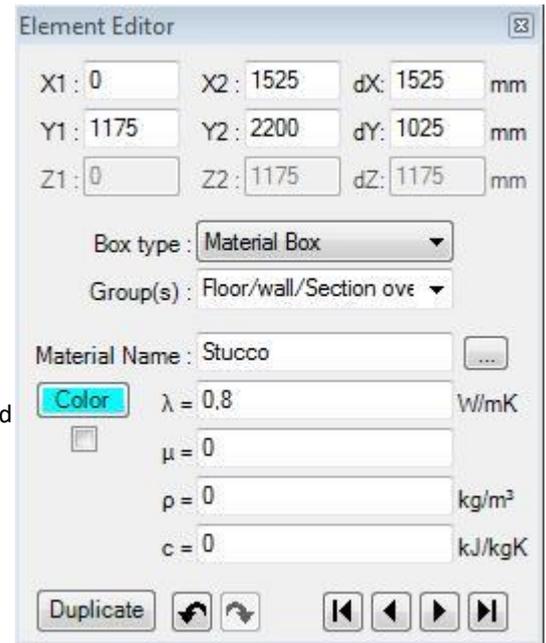
The Element Editor window displays data of a element currently selected for editing in the [element list](#). The selection can be taken within [element selection window](#) and/or in [Elements 2D window](#).

The input of coordinate positions $x_1, x_2, y_1, y_2, z_1, z_2$ of element's box is common to all [element types](#). **Coordinate input is done in millimetres**. The input can also be given by the entry of the first coordinate and of the related thickness interval dX, dY, dZ - the other (second) coordinate will be recalculated accordingly when the input field is being left (e.g.. by pressing the TAB-key).

Also, regardless of [element's type](#), each element can be assigned to one or more [groups](#) for simplifying selections.

By [assigning names to space boxes](#) (or [Power Source boxes](#)) you will define [boundary conditions for the calculation model](#).

During the input [material boxes](#) and [space boxes](#) it is advised to use an earlier prepared [materials list](#) and [surface list](#). These windows "cooperate" with *Element Editor* when used in context of assigning material- or surface properties to the box - a double-click onto a [material's line of materials list](#) (or [surface line of surfaces list](#)) or a drag-drop of one line onto Element Editor window results in an assignment of chosen properties thus eliminating further need of (possibly repetitive) input of those.



Element Editor

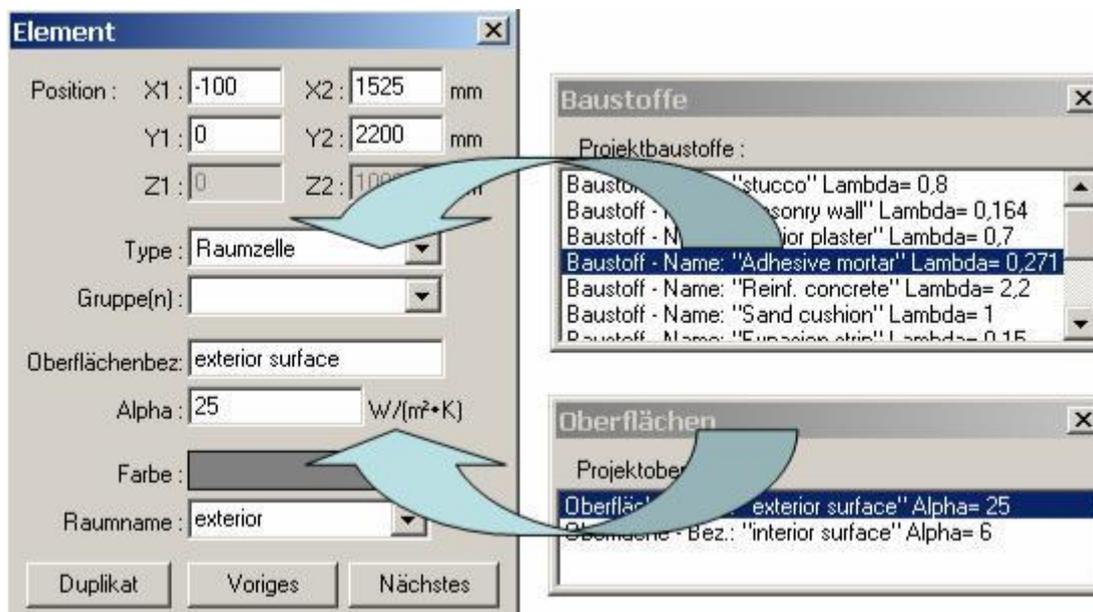
X1: 0 X2: 1525 dX: 1525 mm
 Y1: 1175 Y2: 2200 dY: 1025 mm
 Z1: 0 Z2: 1175 dZ: 1175 mm

Box type: Material Box
 Group(s): Floor/wall/Section ove

Material Name: Stucco

Color $\lambda = 0,8$ W/mK
 $\mu = 0$
 $\rho = 0$ kg/m³
 $c = 0$ kJ/kgK

Duplicate



Both windows (the [materials list](#) or [surfaces list](#)) can be opened by clicking the "..." button placed to the right of material's (or surface's) name field.

Remark: A drag-drop of a material or surface (from a [materials list](#) or [surface list](#)) onto the Element-Editor is only allowed if there are elements of corresponding type (material box or surface box) selected.

Remark: Assigning material properties can be done from the [materials database window](#) too (by a drag-drop or by a double-click).

Pressing a right mouse button over *Element Editor window* exposes (possibly comprehensive) [context menu of element processing](#) which provides further editing functions. This [context menu](#) and functions provided by it are common to all element editing windows and [described separately](#).

Input fields	Are used to input detailed data of one element - see below.
New Duplicate	Inserts a new Element (of the Type "Empty") at the end of the element list (or creates a duplicate of the element currently selected). <u>Remark:</u> When this button is shown focused one can use the ENTER key to activate it instead of the mouse. <u>Remark:</u> Holding SHIFT-key while that button is depressed will additionally swap coordinates of the duplicate.
 Undo(Ctrl-Z)	Undoes (reverts) the latest action(s) in the given editing context .
 Redo(Ctrl-Y)	Redoes (reapplies) the action lately undone in the given editing context .
 First element	Selects the first element in the element list .
 Previous	Selects an element which is previous to the currently selected in the element list .
 Next	Selects an element which is next to the currently selected in the element list .
 Last element	Selects the last element in the element list .
Click right mouse button	Exposes the context menu of element processing functions or group editing functions .
 Button "..."	Clicking onto the button "..." adjacent to material (or surface) name field exposed the window of surfaces or materials .
Double-click onto field label	A double-click over the coordinate field label (left to the input field) will swap the coordinates (value of the first coordinate will be moved to the second field and the one of the second to the first) - eventually useful in context of the Duplicate function. A double-click onto a label of a material or surface property fields (left to the input field) exposes the surfaces window or materials window .
Drag-drop of a material / surface line	A drag-drop of a material or surface (from a materials list or surface list) onto the Element-Editor results in an assignment of chosen properties thus eliminating further need of (possibly repetitive) input of those.

Input fields

Note: You shall switch between input fields by using the TAB-key. Using that key confirms the data entry. On the other hand it provides a good alternative compared to permanent mouse use.

Position x1, x2, y1, y2, z1, z2 Extent dX, dY, dZ	The input of coordinate positions x1,x2,y1,y2,z1,z2 of element's box is common to all element types . Coordinate input is done in millimetres . The input can also be given by the entry of the first coordinate and of the related extent dX, dY, or dZ - the other (second) coordinate will be recalculated accordingly when the input field is being left (e.g.. by pressing the TAB-key). <u>Remark:</u> A double-click over the field label (left to the input field) will swap the coordinates (value of the first coordinate will be moved to the second field and the one of the second to the first) - eventually useful in context of the Duplicate function. <u>Remark:</u> Within a 3D layered model coordinate values of result from the layer's thickness and the sequence of layers. Values of z1 and z2 are automatically calculated and cannot be modified within the Element Editor window. <u>Remark:</u> z1 and z2 have no meaning for 2D model . This values are preset to 0 and 1000 respectively and cannot be modified within the Element Editor window. <u>Remark:</u> If there is some overlapping of element boxes then the ordering of elements in the elements list is significant.
Type	Defines the Type of an element : <ul style="list-style-type: none"> • Empty (EmptyBox, empty element, adiabatic element, deleting/erasing box) • Material (MaterialBox, material element)

	<ul style="list-style-type: none"> • Space (SpaceBox, space element, space air) • Power Source (PowerBox, energy source or sink region) <p>The meaning of various kinds of elements is described further below here.</p> <p><u>Remark:</u> While this input field is shown focused one can use arrow keys or type the first letter of type on the keyboard (E, M, S, P) to select the type from the list.</p>
Group(s)	<p>Enables an assignment of an element to one or more groups.</p> <p><u>Remark:</u> Pressing the right mouse button when the pointer is over this field will expose the context menu of group editing functions.</p>
Specific to material elements:	
Material name	<p>Name of material. It is used for documentation purpose.</p> <p><u>Important:</u> Leading and trailing spaces are significant only during data entry. That white space will be dropped and ignored during the creation of the calculation model. Names differing only by the number of leading or trailing spaces are significant and considered as different materials only during data entry (e.g. Materials list, Data Entry report) but not within Modelling Report nor calculation model.</p>
λ (lambda)	<p>Defines the <i>Thermal Conductivity</i> of the material.</p> <p><u>Remark:</u> Each change to this input field results in automatic colouring of the element (simple inserting of a space character at the end of lambda value followed by pressing TAB key will result in calculation of new colour).</p>
μ (my)	<p>Defines the <i>Water vapour diffusion resistance</i> of the material.</p> <p><u>Remark:</u> A double-click over the field label (left to the input field) will expose materials window.</p>
ρ (rho)	<p>Defines the <i>mass density</i> of the material.</p> <p><u>Remark:</u> A double-click over the field label (left to the input field) will expose materials window.</p>
c (ce)	<p>Defines the <i>heat capacity</i> of the material.</p> <p><u>Remark:</u> A double-click over the field label (left to the input field) will expose materials window.</p>
Colour and AutoColour	<p>Provides the ability to assign a distinctive colour to the element identifying it in graphical views and evaluations.</p> <p>To select the colour click onto the coloured rectangle labelled "Colour" - it will expose <i>colour selection dialog</i>.</p> <p><u>Remark:</u> By default the colour is automatically assigned when change to lambda value is applied, based on predefined colour table.</p> <p>Removing the checkmark will turn the "auto-colour by lambda" application setting off thus assigned colour will be retained during changes to the value of lambda.</p>
Specific to surface elements:	
Surface name	<p>Name of the surface property. It is used for documentation purpose.</p> <p><u>Remark:</u> A double-click over the field label (left to the input field) will expose surfaces window.</p> <p><u>Important:</u> Leading and trailing spaces are significant only during data entry. That white space will be dropped and ignored during the creation of the calculation model. Names differing only by the number of leading or trailing spaces are significant and considered as different surfaces only during data entry (e.g.) Surface list, Data Entry report) but not within Modelling Report nor calculation model.</p>
R_s	<p>Defines the <i>Thermal Transfer Resistance</i> of the surface. This is an alternative input to α.</p> <p><u>Remark:</u> A double-click over the field label (left to the input field) will expose surfaces window.</p>
α (alpha)	<p>Defines the <i>Thermal Transfer Coefficient</i> of the surface. This is an alternative input to R_s.</p> <p><u>Remark:</u> A double-click over the field label (left to the input field) will expose surfaces window.</p> <p>See also: Application setting "Alpha (surface transmittance) hidden"</p>
Space name	<p>Name of the space. It represents distinct boundary condition.</p> <p><u>Remark:</u> The name can be selected from a <i>list of already known space names</i>. While this input field is shown focused the known name can be chosen by using arrow keys or by entering the keyboard key of the first letter in the space name.</p> <p>Important: It is mandatory to assign a distinct name. An empty entry will be automatically replaced by the space named "NONE".</p> <p><u>Important:</u> Leading and trailing spaces are significant only during data entry. That white space will be dropped and ignored during the creation of the calculation model. Names differing only by the number of leading or trailing spaces are significant and considered as different space names only during data entry (e.g. Data Entry report) but not within Modelling Report, calculation model nor within thermalconductance report, results report etc.</p>

Specific to power source elements:	
Power source name	<p>Name of a power source to be assigned to material extent covered by this element. It represents distinct boundary condition.</p> <p><u>Remark:</u> The name can be selected from a <i>list of already known power source names</i>. While this input field is shown focused the known name can be chosen by using arrow keys or by entering the keyboard key of the first letter in the power source name.</p> <p>Important: It is mandatory to assign a distinct name. An empty entry will be automatically replaced by the power source named "NONE".</p> <p><u>Important:</u> Leading and trailing spaces are significant only during data entry. That white space will be dropped and ignored during the creation of the calculation model. Names differing only by the number of leading or trailing spaces are significant and considered as different power source names only during data entry (e.g. Data Entry report) but not within Modelling Report, calculation model nor within thermal conductance report, results report etc.</p>

Element type (kind/type of a element)

The Material Box (material element)

A material box is a extent of a volume filled with some specific material. Heat transport within material is described by [thermal conductivity](#) of material assigned to the box. Vapour transport by diffusion within material is described by its [vapour diffusion resistance number](#). The dynamic, transient behaviour of the material regarding its ability to store heat is given by its [mass density](#) and [heat capacity](#).

Remark: The [equivalent heat conductivity of small air cavities and air gaps](#) can be calculated with the [air cavity calculator tool](#).

The Space Box (space element, space air)

A space box represents an air filled room of some defined temperature. The heat transport occurs only at surfaces of material boxes connected to the space.

All *space boxes* assigned same space name are merged together as one space. Each such space will later be assigned to one [temperature \(the boundary condition\)](#) during the simulation and evaluation. By consequently choosing respective space names – e.g. exterior, room, kitchen, bath – or – e.g.. Room 0, Room 1, ... - one creates a list of [boundary conditions](#) for simulations.

Important: It is mandatory to assign a distinct name. An empty name will be automatically replaced by the space named "NONE".

The Power Source (PowerBox, power element, energy source or sink region/area)

A power source generates no new volume box. Contrary to material boxes it only defines an area (cube) of power source or power sink. All material boxes covered by this are (or their parts) will be assigned to that power source and will receive heat energy from this source.

All *power source boxes* assigned same name are grouped together as one power source. Each such source will later be assigned to one [power density \(the boundary condition\)](#) during the simulation and evaluation. By consequently choosing respective power source names – e.g. heating interlayer, wall heating, power stick – or – e.g.. PS 0, PS 1, ... - one creates a list of [boundary conditions](#) for simulations.

Important: It is mandatory to assign a distinct name. An empty name will be automatically replaced by the power source named "NONE".

The Empty box (EmptyBox, empty element, adiabatic element, deleting/erasing/adiabatic box)

Empty boxes are primarily used to excavate or cut areas of the construction creating **adiabatic** boundaries. A metaphor to „Vacuum“, without any kind of heat transport, can be helpful.

Group(s) of Elements

Each one element can be assigned to one or more group names. Elements can then be quickly selected by the group name - see [element list](#) and the [context menu of element processing functions](#).

The assignment of an element to some group is done by entering the group name into the group name input field. Names of groups the element is already assigned to can be listed by clicking onto the arrow symbol adjacent to the input field. While this input field is shown focused one can use arrow keys to browse through the list.

The click with the right mouse button of the input field *Group(s)* exposes the *context menu of group editing functions*.



Remove from Group	The Element will be removed from the group actually displayed in the input field
--------------------------	--

Boundary conditions

All [space boxes](#) merge by their name together to spaces. Every space is assigned one temperature (the boundary condition) during the simulation. By consequently choosing respective space names – e.g. exterior, room, kitchen, bath – or – e.g.. Room 0, Room 1, ... - one creates a list of [boundary conditions](#) for simulations.. Same applies to [power sources](#).

All spaces (their temperatures) and all power sources (their power densities) provide boundary conditions for results calculated by the program collectively. The name of a space or a power source uniquely identifies such boundary condition which is then further processed and used during [Evaluation](#).

If the particular space- or power source name is already known one can click onto the arrow adjacent to the name input field and select it directly from the list exposed. While the name input field is shown focused one can use arrow keys to select the name or type the first letter of the name in the list.

Important: It is mandatory to assign distinct names for spaces and power sources. An empty name will be automatically replaced by the power source named "NONE".

Important: The component prepared for the calculation **must contain two boundary conditions at least** - see: [Solver window - data integrity and messages](#).

Element Editor

X1: 0 X2: 1525 dX: 1525 mm
 Y1: 1175 Y2: 2200 dY: 1025 mm
 Z1: 0 Z2: 1175 dZ: 1175 mm

Box type: Empty Box
 Group(s): Floor/wall/Section over

Duplicate      

Convert to Slope/Roundness

Element Editor

X1: -100 X2: 1525 dX: 1625 mm
 Y1: 0 Y2: 2200 dY: 2200 mm
 Z1: 0 Z2: 1175 dZ: 1175 mm

Box type: Space Box
 Group(s): Floor/wall/Section over

Surface Name: Ext. transfer coeff. ...

Color

Rs (Heat flow) = 0,04 m²K/W
 Rs (Temperature) = 0,04 m²K/W

Space Name: Room 0

Duplicate      

Convert to Slope/Roundness

Element Editor

X1: 349 X2: 1525 dX: 1176 mm
 Y1: 1250 Y2: 1310 dY: 60 mm
 Z1: 0 Z2: 1175 dZ: 1175 mm

Box type: Power Box
 Group(s): Floor/wall/Section over

Power Name: PS 0

Duplicate      

Convert to Slope/Roundness

See also: [Element selection window](#), [Elements 2D](#), [Elements 3D](#), [Materials window](#), [Surfaces window](#), [Element kinds/types](#), [Context menu of element processing](#), [Colour settings file LambdaToColor.ColorList](#), [Coordinate system](#), [Air Cavity Calculator \(Tool\)](#)

Context menu of element editing

The *context menu of element editing* can be exposed from various [editing windows](#) when the *right mouse button* is depressed.

The set of functions supplied within this context menu depends on the actual number of elements selected from the [element list](#).

Edit Element. (Ctrl- E)	Exposes the Element Editor window .
Materials&Surfaces (Ctrl-M)	Exposes the Materials and/or Surfaces window.
Copy (CTRL-C)	Creates a copy of currently selected element(s) and places it onto the clipboard. Element(s) can then be pasted from it See also: Paste
Cut (Strg-X)	Currently selected elements are deleted from the element list, but their copies are placed onto the clipboard. Element(s) can then be pasted from it. See also: Paste , Delete
Paste (Strg-V)	Element(s) currently on the clipboard are insert just beyond the one currently selected in the element list (or appended to the end of element list if none or several elements are selected). After being inserted all these elements are marked selected thus being ready for further processing (Translate , Rotate , Mirror etc.). See also: Duplicate

Edit Element	Ctrl+E
Materials & Surfaces	Ctrl+M
Copy	Ctrl+C
Cut	Ctrl+X
Paste	Ctrl+V
Add	▶
Duplicate	
Move (Order)	▶
Delete	Del
Translate	
Stretch	
Rotate	
Mirror	
Assign	▶
Remove from Group	▶
Select	▶
Undo	Ctrl+Z
Redo	Ctrl+Y

Add ▶	Append
	Before Selected
	After Selected

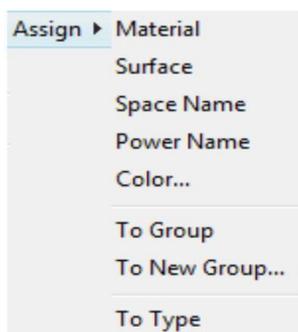
Add→Append	Appends a new element (of type "Empty") to the end of element list
Add→Before Selected	Inserts a new element (of type "Empty") just before the one currently selected into the element list .
Add→After Selected	Inserts a new element (of type "Empty") just after the one currently selected into the element list .
Duplicate	Inserts a copy of currently selected elements into the element list . Remark: Holding SHIFT-key while that menu is activated will additionally swap coordinates of the duplicate. If several elements are selected their copies are appended to the end of element list retaining their order. After being inserted all these element copies are marked selected thus being ready for further processing (Translate , Rotate , Mirror etc.). See also: Copy & Paste also Translate window

Move (Order)→Relegate to Backmost	Reorders selected element(s) in the element list to the "backmost" position. The element is then located at the beginning of the list thus now being potentially overlapped by all other elements. If more elements are selected then each one is relegated by the same distance as the backmost selected one. This action is inhibited if the chosen element does already occupy the first (backmost) location in the list.
Move (Order)→Relegate	Reorders selected element(s) in the element list one position "backwards". The element is then located earlier in the list thus now being potentially overlapped by another element which have thus been promoted to the location of the element relegated. If more elements are selected then each one is relegated by one position in the list order. This action is inhibited if the chosen element does already occupy the first (backmost) location in the list.

Move (Order)→Promote	Reorders selected element(s) in the element list one position "towards front". The element is then located later in the list thus now potentially overlapping another element which have thus been relegated to the location of the element promoted. If more elements are selected then each one is promoted by one position in the list order. This action is inhibited if the chosen element does already occupy the last (front most) location in the list.
Move (Order)→Promote to Frontmost	Reorders selected element(s) in the element list to the "frontmost" position. The element is then located at the end of the list thus now potentially overlapping all other elements. If more elements are selected then each one is promoted by the same distance as the frontmost selected one. This action is inhibited if the chosen element does already occupy the last (front most) location in the list.
Delete (DEL key)	Deletes (irrevocably) currently selected element(s) from the element list . After deletion of single element is executed the selection moves to a foregoing element. When first element is deleted the selection move to the element advanced to first position. Deletion request of one or more elements is executed after being additionally confirmed in a dialog window shown. See also: Cut

Translate	Provides a translation of one or many elements by a distance (dX, dY, dZ) entered in the window Translate . <u>Remark:</u> Translation itself is invoked from the Translation window .
Stretch	Provides stretching of one or many elements by a specified distance in specified coordinate direction. Extension (or shrinking) of elements is given by specific plane coordinate selection an shift interval within the Stretch window .
Rotate	Provides rotation of one or more elements around an axis parallel to one of coordinate axes. The position of the rotation axis is given by the choice of one rotation plane and two coordinates within the window Rotate .
Mirror	Provides mirroring of one or more elements to a plane parallel to one of coordinate planes. The position of mirroring plane is given by the choice of mirroring direction and its coordinate position in the window Mirror .

Select→By Group→...	Selects all elements assign to one specific Group (the group name) . <u>Remark:</u> The menu offers a list of all available groups to chose from. If there are no groups used yet this menu has no function. <u>Remark:</u> For 3D-Layered models the selection operation is applies only to currently active layer.
Select→By Type→...	Selects all element of one specific type (Empty, Material, Space, etc.) <u>Remark:</u> For 3D-Layered models the selection operation is applies only to currently active layer.
Select→By Name→...	Selects all elements having one specific name. <u>Remark:</u> The menu offers a list of all names - names of spaces, power sources, surfaces and materials) to chose from. <u>Remark:</u> For 3D-Layered models the selection operation is applies only to currently active layer.
Select→All (CTRL-A)	Select all elements. <u>Remark:</u> For 3D-Layered models the selection operation is applies only to currently active layer.



Assign→Material→...	Assign a material to all selected material boxes. All materials used in the project so far are offered for choice. <u>Remark:</u> If within current selection there are elements of type incompatible with the assignment to be performed, the action is executed on compatible elements only after additional user confirmation dialog. If several elements are selected user's confirmation is also requested by a dialog. <u>Error message</u> is shown if neither element matches the expected type.
Assign→Surface→...	Assign a surface to all selected space boxes. All surfaces used in the project so far are offered for choice. <u>Remark:</u> If within current selection there are elements of type incompatible with the assignment to be performed, the action is executed on compatible elements only after additional user confirmation dialog. If several elements are selected user's confirmation is also requested by a dialog. <u>Error message</u> is shown if neither element matches the expected type.
Assign→Space name→...	Assign a space name to all selected space boxes. All space names used in the project so far are offered for choice. <u>Remark:</u> If within current selection there are elements of type incompatible with the assignment to be performed, the action is executed on compatible elements only after additional user confirmation dialog. If several elements are selected user's confirmation is also requested by a dialog. <u>Error message</u> is shown if neither element matches the expected type.
Assign→Power name→...	Assign a power source name to all selected power source boxes. All power source names used in the project so far are offered for choice. <u>Remark:</u> If within current selection there are elements of type incompatible with the assignment to be performed, the action is executed on compatible elements only after additional user confirmation dialog. If several elements are selected user's confirmation is also requested by a dialog. <u>Error message</u> is shown if neither element matches the expected type.
Assign→Colour...	Assign a colour to all selected element (standard colour dialog is shown) See also: Element Editor
Assign→To Group→...	All selected elements are assigned to the group name chosen (in addition to group assignments already in place). <u>Remark:</u> The menu offers a list of all groups used in the project so far to chose from. If there are no groups used yet this menu has no function.
Assign→To New Group...	A dialog will be offered to enter the group name. All selected elements are assigned to the group name entered (in addition to group assignments already in place).
Assign→To Type→...	The type of currently selected element(s) will be changed to the one of <ul style="list-style-type: none"> • Matrial box(es), • Power box(es), • Space box(es) or • Empty box(es) When the assignment is done for multiple elements a confirmation will be required.
Remove from Group	Assignment to one specific group name is deleted from all selected elements. <u>Remark:</u> The menu offers a list of all groups applicable to selected elements to chose from. If no one of selected elements is assigned to any group this menu has no function.
Undo (Ctrl-Z)	Undoes (reverts) the latest action(s) in the given editing context .
Redo(Ctrl-Y)	Redoes (reapplies) the action lately undone in the given context

Important: Shortcuts shown above *CTRL-A*, *CTRL-C*, *CTRL-X*, *CTRL-V*, *DEL* of this context menu are not available with [Element-Editor window](#). These key combinations have further specific functions respective to input fields of Element-Editor window.

See also: [Element Editor](#), [Element selection window](#), [Elements 2D window](#), [Elements 23 window](#), [Editing windows \(Input windows\)](#)

3D-Layer selection window

Layer selection window displays a list of all 3D layers of a construction entered as a [3D Layered project](#).

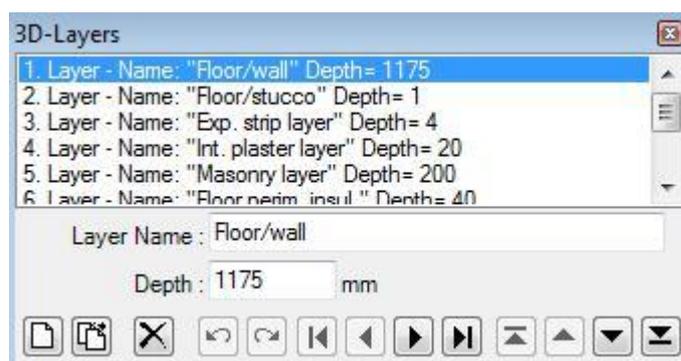
Important: The [order of layers](#) in the list is significant for the resulting [3D construction](#).

From this window one layer can be selected for editing, new layers can be inserted, the order of layers can be changed (moved towards front or back) or layers can be deleted from the project.

The layer currently processed is shown as selected. Data of the selected 3D-layer is shown and can be modified within [editing part of the window](#)

Remark: Only one layer can be selected for processing.

Remark: The windows [Element selection \(browser\)](#) and [Elements 2D](#) always display elements of the currently selected layer.



Layer list	List of all input layers in the specific order along the Z direction of coordinate system
Layer Name	Name of the layer. It is used for documentation purpose.
Depth	Define the thickness of the layer in the direction of Z axis.
 New	Appends a new layer (with uniform thickness of 1000mm) onto the end of layer list
 Duplicate	Creates an exact copy of selected layer (including copies of elements contained in this layer) and inserts the new layer into the layer list just beyond the selected one.
 Delete	Deletes the currently selected layer from the list. Warning: all elements which belong to this layer are inadvertently deleted also. Remark: You will be asked for confirmation.
 Undo(Ctrl-Z)	Undoes (reverts) the latest action(s) in the given editing context .
 Redo(Ctrl-Y)	Redoes (reapplies) the action lately undone in the given editing context .
	Selects First, Previous, Next or Last layer.
 Relegate to Backmost	Reorders the currently selected layer to the backmost position in the list. The layer is then shown as first in the list.
 Relegate	Reorders the currently selected layer one position "towards back" in the list. The layer is then shown earlier in the list.
 Promote	Reorders the currently selected layer one position "towards front" in the list. The layer is then shown later in the list.
 Promote to Frontmost	Reorders the currently selected layer to the frontmost position in the list. The layer is then shown as last in the list.
Click right mouse button	Exposes the context menu of layer editing functions
Click left mouse button	Selection of a layer from the layer list
Double click left mouse button	Selection of a layer from the list and exposure of Element selection (browser) and Elements 2D windows.

Layer list

Each line in the list represent one layer in the model input. Adjacent to layer's name its thickness is displayed. The windows [Element selection \(browser\)](#) and [Elements 2D](#) always display elements of the currently selected layer. The construction resulting from current modelling can be observed within the [Elements 3D window](#).

The [order of layers](#) in the list corresponds to the sequence of layers as they are stacked together along Z coordinate direction. See the description of the [project type "3D layered project"](#).

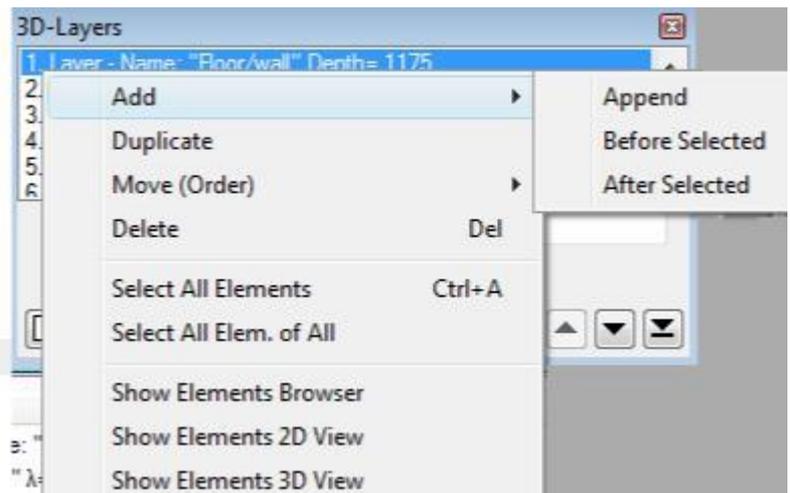
Note: You shall switch between input fields by using the TAB-key. Using that key confirms the data entry. On the other hand it provides a good alternative compared to permanent mouse use.

Layer editing

Data of a selected layer is displayed within [layer editing part of the window](#) for thickness adjustments.

A double click onto the layer's line exposes the windows [Element selection \(browser\)](#) and [Elements 2D](#) which display elements of the currently selected layer for further processing.

When the right mouse button is presses over layer list a [context menu of layer editing](#) is shown and offers further layer processing functions.



Layer selection

A click onto layer's line in the layer list selects corresponding layer for further processing.



Selection can be done by arrow keys or by clicking the buttons Next/Prev. also.

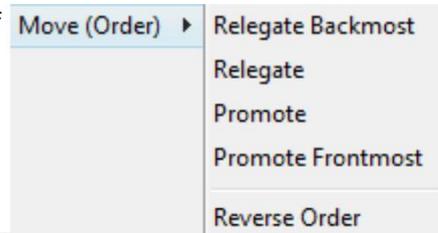
Reordering layers

The order in which layers are shown in the list is significant for the stacking order of these layers in Z coordinate direction. Reordering of layers results in different stacking order.



Buttons "Relegate" and "Promote" are used to move selected layer within the list and thus change the stacking order of layers.

See the description of the [project type "3D layered project"](#).



Functions of the context menu

Add→Append	Appends a new layer (with uniform thickness of 1000mm) onto the end of layer list.
Add→Before Selected	Inserts a new layer (with uniform thickness of 1000mm) just before the one selected in the list.
Add→After Selected	Inserts a new layer (with uniform thickness of 1000mm) just below the one selected in the list.
Duplicate	Creates an exact copy of selected layer (including copies of elements contained in this layer) and inserts the new layer into the layer list just beyond the selected one.
Move (Order)→Relegate to Backmost	Reorders the currently selected layer to the backmost position in the list. The layer is then shown as first in the list.
Move (Order)→Relegate	Reorders the currently selected layer one position "towards back" in the list. The layer is then shown earlier in the list.
Move (Order)→Promote	Reorders the currently selected layer one position "towards front" in the list. The layer is then shown later in the list.

Move (Order)→Promote to Frontmost	Reorders the currently selected layer to the frontmost position in the list. The layer is then shown as last in the list.
Move (Order)→Reverse Order	Reorders elements of the list to the opposite direction. The earlier frontmost layers displays as backmost then.
Delete	Deletes the currently selected layer from the list. Warning: all elements which belong to this layer are inadvertently deleted also.
Select All Elements (Ctrl-A)	Selects all elements of the current layer for further processing. Remark: Use it to emphasize layer within transparent Elements3D view or as preparation for translation/rotation etc.
Select All Elements in All Layers	Selects all elements in all layers for further processing.
Show Elements Browser	Exposes the Element selection (browser) - window showing elements of currently selected 3D-layer.
Show Elements 2D View	Exposes the Elements 2D - window showing elements of currently selected 3D-layer.
Show Elements 3D View	Exposes the Elements 3D window .

Remark: Layer list is only meaningful for [3D Layered Project](#). **Within** a 2D-Model editing of layer list is inhibited (2D model corresponds to only one layer with fixed 1 meter thickness). Within 3D projects this window cannot be used because a 3D project contains only one ordered list of elements and not layers.

See also: [Element selection window](#), [Elements 2D window](#), [Elements 3D window](#), [Project types](#), [The component](#)

Layer Editor window

Layer Editor window is used to edit properties of one layer within a 3D layered project.

This window is shown within [3D-layer selection window](#).



Layer Name : Floor/wall
Depth : 1175 mm

Layer Name	Name of the layer. It is used for documentation purpose.
Depth	Define the thickness of the layer in the direction of Z axis.

Note: You shall switch between input fields by using the TAB-key. Using that key confirms the data entry. On the other hand it provides a good alternative compared to permanent mouse use.

Remark: Layer is only meaningful for [3D Layered Project](#). **Within** a 2D-Model editing of layer is inhibited (2D model corresponds to only one layer with fixed 1 meter thickness). Within 3D projects this window cannot be used because a 3D project contains only one ordered list of elements and not layers.

See also: [Layer selection window](#), [Coordinate system](#)

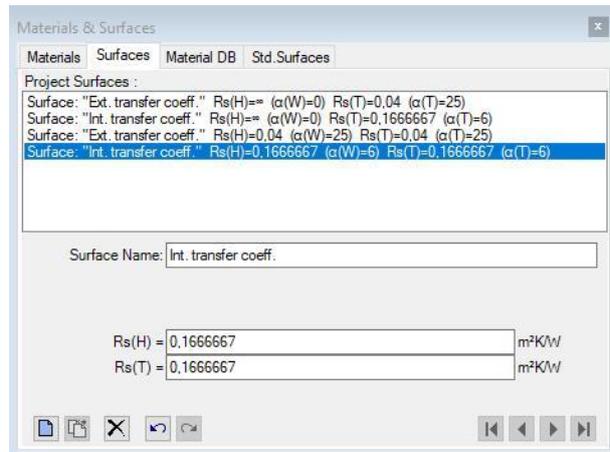
Now it is possible to change the selected material in all layers the layer modeling.

Select a material you want to change in the window "Material&Surface" , then click with the left mouse button on the new button (right).

The material master data will open.

Double-click to adopt a new material. Right mouse button - selected material - exchange all elements.

Materials & Surfaces window



lists of Materials & Surfaces window is used to maintain project specific material and surface properties. It is used as an entry point to following editing windows:

- [Materials window](#)
- [Surfaces window](#)
- [Material Database window](#)
- [Standard Surfaces window](#)

The primary use of this window is to prepare lists of materials and surfaces which, in cooperation with other windows, allow significant simplification and acceleration during the input of element properties for material and space boxes.

To expose this window select it from the [menu View->Data Input & Entry](#) or double click onto the respective material or surface property input field or its label in [Element Editor \(conductivity, diffusion resistance, transfer coefficient or transfer resistance\)](#).

Remark: Insertion of materials can be performed via *Drag & Drop* from [Materials Database window](#). The [Material Database](#) can be opened by activating the Tab "Material DB".

Remark: The Tab "Std.Surfaces" offers standardized, non editable, surface properties (normative values).

See also: [Materials window](#), [Surfaces window](#), [Material Database window](#), [Element Editor](#), [Elements 2D](#), [Element list](#), [Material Editor](#), [Surface Editor](#)

Materials window

The Materials window is used to create and maintain the list of project specific material properties.

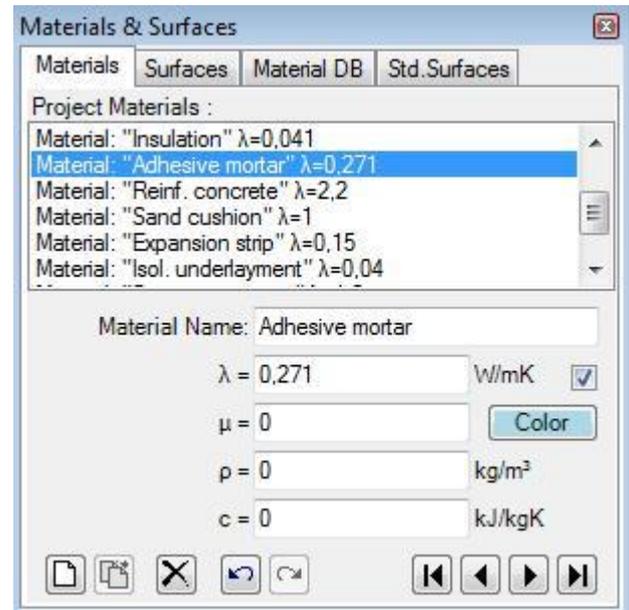
The list of materials offered in this window can be [synchronised](#) with materials directly entered and used in the model, thus all materials constituting the component are displayed here also. Materials can be [merged from other projects](#) and combined in this list.

The primary use of this window is to prepare lists of materials which, in cooperation with other windows, allow significant simplification and acceleration during the input of element properties for material boxes. Materials already prepared within another project can be [merged](#) to the list also.

Upper part of the window lists [materials](#). The bottom part - [Material Editor](#) - allows definition of material properties.

Important: Changes to material properties in the project specific materials list do not automatically propagate to properties of construction elements. Applying new material properties to selected elements must be done explicitly - either by a double-click onto a list item, via a drag-drop operation from the material list or by [the bulk replacement of properties](#).

Remark: Additional materials can be also selected from [Material Database window](#) by *drag-drop* from database ([Material Database](#) can be opened from the Tab "Material DB"). Dropping some material identical to the one in the list will be inhibited.



Material list

A double-click onto a line representing material in this window results in an assignment of material properties of selected material to currently selected material elements (selected from [elements list](#)).

Also, by *drag-drop operation* (by dragging with the mouse), a material line can be dragged onto [element editor](#) window resulting in assignment of material properties to the element currently shown there.

Materials window can be opened either from [main menu_View→Data Input & Entry→Materials](#) or from [element editor](#) by a *double-click onto material name, conductivity and other material property input fields*.

Double-click left mouse button	Transfer of material properties of selected material onto currently selected element(s) in the element list .
Dragging with left mouse button	Transfer of material properties of dragged material to material element currently edited in element editor , or: Transfer of material properties of dragged material to currently selected material element(s) in the element list .
Click right mouse button	Exposes the context menu of material editing functions .
Click left mouse button	Selection of material from material list. Properties of selected material are displayed in Material Editor for further processing.

Remark: If there are several elements selected (from [element list](#)) then a double-click onto the material line (or drag-drop of material line) result in the assignment of material properties to all selected elements. Such bulk element assignment has to be confirmed by the user. If within selected elements there are some of incompatible type (non material) further user confirmation is requested. Error message is shown if neither element matches the expected type.

 New	Appends a new material onto the end of the material list
 Delete	Deletes the currently selected material from the list. Remark: The material is deleted from the material list only.
 Undo(Ctrl-Z)	Undoes (reverts) the latest action(s) in the given editing context .
 Redo(Ctrl-Y)	Redoes (reapplies) the action lately undone in the editing context .
	Selects First, Previous, Next or Last material.

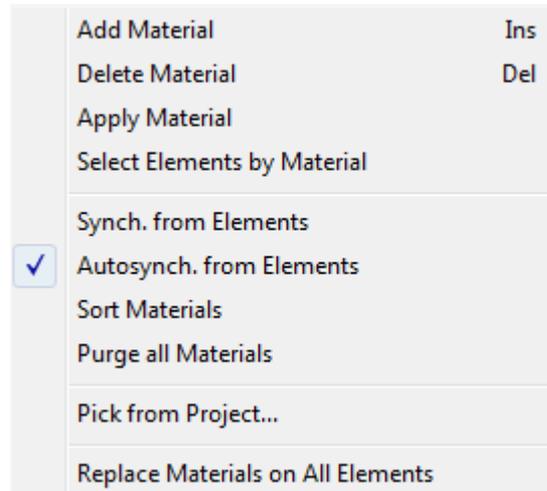
Material selection

A click onto material's line in the material list selects corresponding material for further processing.

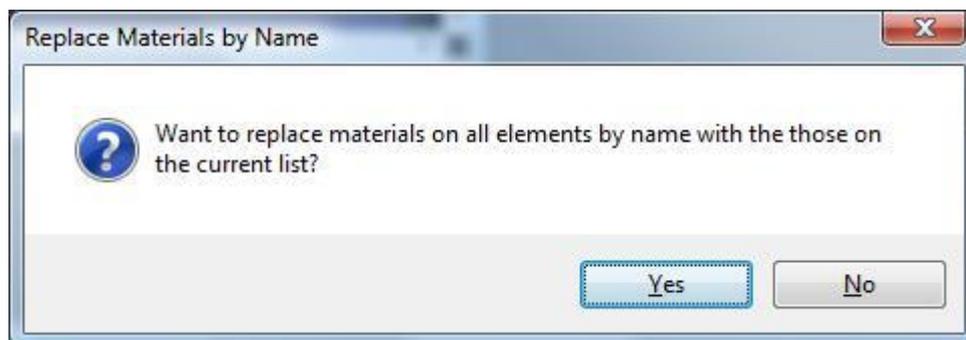


Selection can be done by arrow keys or by clicking the buttons Next/Prev. also.

Funktion of context menu



Add Material INS-Key	A new material is appended at the end of the list.
Delete Material DEL-Key	Deletes selected material from the list.
Apply Material	Transfer material properties of a selected material onto currently selected element(s) in the element list . Remark: Visible when exactly one surface is selected.
Select Elements by Material	Selects all elements in the element list having one specific material assigned. Remark: Visible when exactly one surface is selected. Remark: For 3D-Layered models the selection operation applies only to currently active layer.
Synch. from Elements	Executes immediate synchronisation of material list from properties of all elements currently in the modelled component
Autosynch. from Elements	The synchronisation of material list from properties of all elements in the modelled component will be executed automatically (checkmark is visible) and can be turned on or off here.
Sort Materials	Materials will be sorted by <i>name</i> , λ , μ , ρ and <i>c</i> .
Purge all Materials	Removes all materials from the list
Pick from Project...	Materials saved within another project will be read and appended to the list. Remark: After importing the data from another project the (optional) bulk replacement of material properties on all elements with those just imported will be offered.
Replace Materials on All Elements	The material properties of all elements will be replaced by name with those from the current material list (bulk replacement). Remark: Shall duplicate name appear in the list only the first instance in the order will be applied. Remark: For 3D-Layered models the operation applies only to currently active layer.



Synchronisation of material list

The list of materials offered in this window can be synchronized (extended) with materials currently in use in the modelled component, thus all materials used in the model are also displayed. This provides the ability to transfer properties directly entered in [element editor](#) to the list and have them at hand as additional entries.

Synchronisation can be explicitly requested from the [context menu](#) of the material list ("Synch. from Elements") or will automatically be executed on each [element selection change](#) when the [application setting "Autosynch. from Elements"](#) is active.

Synchronisation executes by following steps:

- All elements of modelled component are analysed.
- Material properties of each material element are extracted.
- If no such material (name, lambda, my) can be found in material list it is appended to it.
- Materials appended during this synchronisation run and appended at the end of the list are sorted alphabetically.

This procedure assures, that entries are retained in the list and only "missing" materials will extend it.

Remark: Automatic synchronisation of list contents, if active, occurs on each [element selection change](#). Changes to element properties (in [Element Editor](#)) alone do not result in automatic synchronisation yet. Immediate synchronisation can be explicitly requested from the [context menu](#) of the material list ("Synch. from Elements").

Tip: To have the list showing only materials actually used in the modelled component delete all materials from the list (this can be quickly done with DEL-key) and then request immediate synchronisation from component's elements. Unused materials are not removed automatically from the list!

Remark: Even if values of ρ (density) and c (heat capacity) are transferred from material database, these values cannot be modified unless the [licence feature HARMONIC or TRANSIENT](#) is available (these data is not used during nor required for the steady state simulation).

See also: [Element Editor](#), [Elements 2D](#), [Element list](#), [Material Editor](#)

Material Editor window

Material Editor is used to maintain properties of material. This window is displayed as part of [materials window](#).

Material Name	Name of material. It is used for documentation purpose.
λ (lambda)	<i>Thermal Conductivity</i> of the material.
μ (mue)	<i>Water vapour diffusion resistance</i> of the material.
ρ (rho)	<i>Mass density</i> of the material.
c (ce)	<i>Heat capacity</i> of the material.
Colour and AutoColour	Provides the ability to assign any distinctive colour to the element identifying it in graphical views and evaluations. To select the colour click onto the coloured rectangle labelled "Colour" - it will expose <i>colour selection dialog</i> . Remark: By default the colour is automatically assigned when change to lambda value is applied, based on predefined colour table . Removing the checkmark will turn the " auto-colour by lambda " application setting off thus assigned colour will be retained during changes to the value of lambda
New	Adds a new entry to the material list and selects it for editing.
Delete	Deletes the currently selected material from material

The screenshot shows the Material Editor window with the following fields and controls:

- Material Name: Adhesive mortar
- λ = 0,271 W/mK (with a checkmark icon)
- μ = 0 (with a "Color" button)
- ρ = 0 kg/m³
- c = 0 kJ/kgK
- Bottom toolbar: Includes icons for New, Delete, Undo, Redo, and navigation arrows.

	list .
Prev.	Selects previous material in the material list for editing.
Next	Selects next material in the material list for editing

Important: Changes to material properties in the project specific [material list](#) do not automatically propagate to properties of construction elements. Applying new material properties to selected elements must be done explicitly - either by a double-click onto a [material list](#) item or via a drag-drop operation from [material list](#) .

Remark: The [equivalent heat conductivity of small air cavities and air gaps](#) can be calculated with the [air cavity calculator tool](#).

Note: You shall switch between input fields by using the TAB-key. Using that key confirms the data entry. On the other hand it provides a good alternative compared to permanent mouse use.

See also: [Materials window](#), [Air Cavity Calculator \(Tool\)](#)

Surfaces window

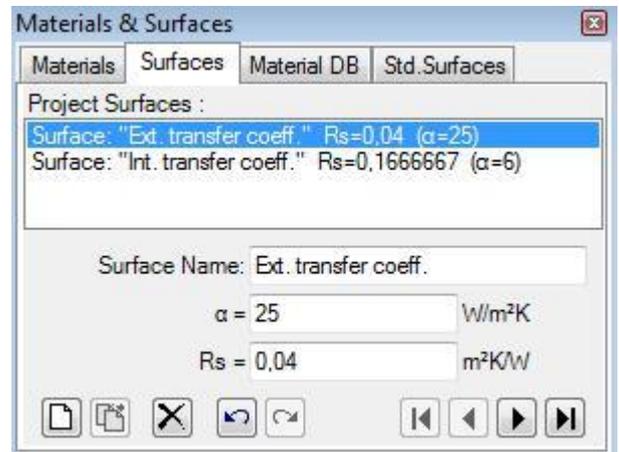
The Surfaces window is used to create and maintain the list of project specific surface properties.

The list of surfaces offered in this window can be [synchronised](#) with surfaces directly entered and used in the model, thus all surfaces constituting the component are displayed here also.

Surfaces can be [merged from other projects](#) and combined in this list.

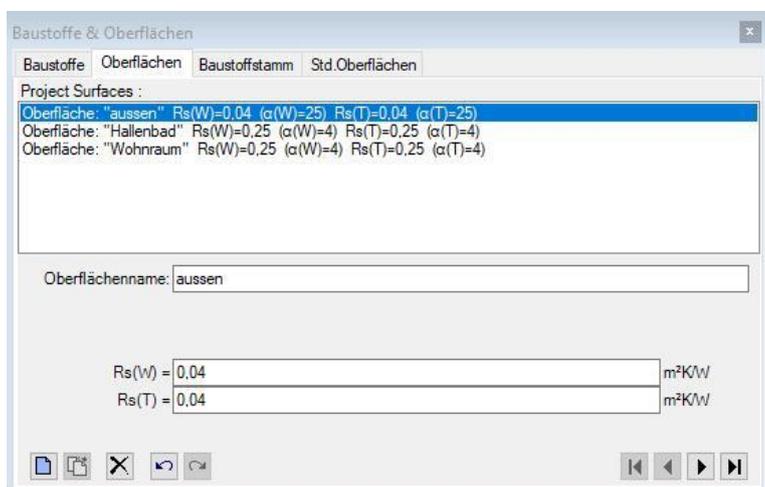
The primary use of this window is to prepare lists of surfaces which, in cooperation with other windows, allow significant simplification and acceleration during the input of element properties for space boxes.

Upper part of the window [lists surfaces](#). The bottom part - [Surface Editor](#) - allows definition of surface properties.



Important: Changes to surface properties in the project specific surface list do not automatically propagate to properties of construction elements. Applying new surface properties to selected elements must be done explicitly - either by a double-click onto a list item, via a drag-drop operation from the surface list or by [the bulk replacement of properties](#).

Remark: The [tab "Std.Surfaces"](#) provides a list of (read only) [normative surface properties](#).



Surface list

A double-click onto a line representing surface in this window results in an assignment of surface properties of selected surface to currently selected space elements (selected from [elements list](#)).

Also, by *drag-drop operation* (by dragging with the mouse), a surface line can be dragged onto [element editor](#) window resulting in assignment of surface properties to the element currently shown there.

Surfaces window can be opened either from [main menu View→Data Input & Entry→Surfaces](#) or from [element editor](#) by a *double-click onto surface name, transfer coefficient or resistance input fields*.

Surface list	List of all surface properties entered
Double-click left mouse button	Transfer of surface properties of selected material onto currently selected elemen(s) in the element list .
Dragging with left mouse button	Transfer of surface properties of dragged surface to space element currently edited in element editor , or: Transfer of surface properties of dragged surface to currently selected space elemen(s) in the element list .
Click right mouse button	Exposes the context menu of surface editing functions .
Click left mouse button	Selection of surface from surface list. Properties of selected surface are displayed in Surface Editor for further processing.

Remark: If there are several elements selected (from [element list](#)), then a double-click onto the surface line (or drag-drop of surface line) result in the assignment of surface properties to all selected elements. Such bulk element assignment has to be confirmed by the user. If within selected elements there are some of incompatible type (non space) further user confirmation is requested. Error message is shown if neither element matches the expected type.

 New	Appends a new surface onto the end of the surface list
 Delete	Deletes the currently selected surface from the list. Remark: The surface is deleted from the surface list only.
 Undo(Ctrl-Z)	Undoes (reverts) the latest action(s) in the given editing context .
 Redo(Ctrl-Y)	Redoes (reapplies) the action undone in the given editing context .
	Selects First, Previous, Next or Last surface.

Surface selection

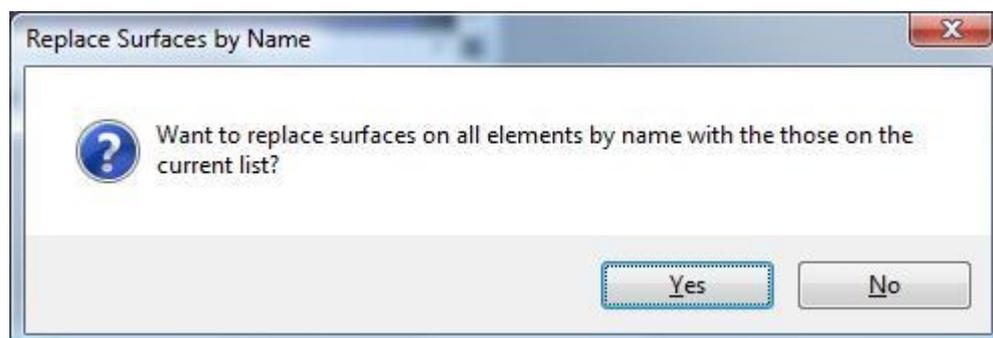
A click onto surface's line in the surface list selects corresponding surface for further processing.



Selection can be done by arrow keys or by clicking the buttons Next/Prev. also.

Functions of the context menu

Add Surface Ins Delete Surface Del Apply Surface Select Elements by Surface <hr/> Synch. from Elements <input checked="" type="checkbox"/> Autosynch. from Elements Sort Surfaces Purge all Surfaces <hr/> Pick from Project... <hr/> Replace Surfaces on All Elements	A new surface is appended at the end of the list.
Add Surface INS-Key	
Delete Surface DEL-Key	Deletes selected surface from the list.
Apply Surface	Transfer surface properties of a selected surface onto currently selected element(s) in the element list . <u>Remark:</u> Visible when exactly one surface is selected.
Select Elements by Surface	Selects all elements in the element list having one specific surface assigned. <u>Remark:</u> Visible when exactly one surface is selected. <u>Remark:</u> For 3D-Layered models the selection operation applies only to currently active layer.
Synch. from Elements	Executes immediate synchronisation of surface list from properties of all elements currently in the modelled component
Autosynch. from Elements	The synchronisation of surface list from properties of all elements in the modelled component will be executed automatically (checkmark is visible) and can be turned on or off here.
Sort Surfaces	Surfaces will be sorted by name and alpha.
Purge all Surfaces	Removes all surfaces from the list
Pick from Project...	Surfaces saved within another project will be read and appended to the list. <u>Remark:</u> After importing the data from another project the (optional) bulk replacement of surface properties on all elements with those just imported will be offered.
Replace Surfaces on All Elements	The surface properties of all elements will be replaced by name with those from the current surface list (bulk replacement). <u>Remark:</u> Shall duplicate name appear in the list only the first instance in the order will be applied. <u>Remark:</u> For 3D-Layered models the operation applies only to currently active layer.



Synchronisation of surface list

The list of surfaces offered in this window can be synchronized (extended) with surfaces currently in use in the modelled component, thus all surfaces used in the model are also displayed. This provides the ability to transfer properties directly entered in [element editor](#) to the list and have them at hand as additional entries.

Synchronisation can be explicitly requested from the [context menu](#) of the material list ("Synch. from Elements") or will automatically be executed on each [element selection change](#) when the [application setting "Autosynch. from Elements"](#) is active.

Synchronisation executes by following steps:

- All elements of modelled component are analysed.
- Surface properties of each space element are extracted.
- If no such surface (name, α , R_s) can be found in surface list it is appended to it.
- Surfaces appended during this synchronisation run and appended at the end of the list are sorted alphabetically.

This procedure assures, that entries are retained in the list and only "missing" surfaces will extend it.

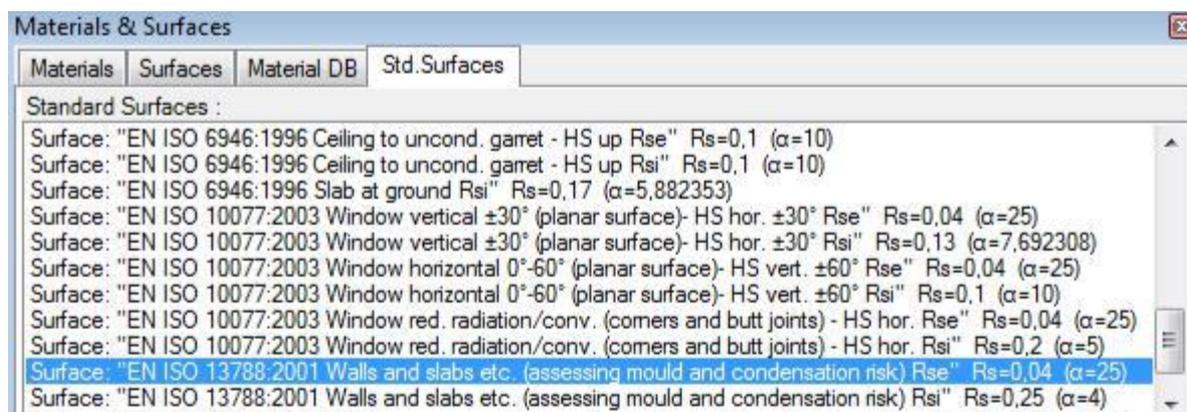
Remark: Automatic synchronisation of list contents, if active, occurs on each [element selection change](#). Changes to element properties (in [Element Editor](#)) alone do not result in automatic synchronisation yet. Immediate synchronisation can be explicitly requested from the [context menu](#) of the surface list ("Synch. from Elements").

Tip: To have the list showing only surfaces actually used in the modelled component delete all surfaces from the list (this can be quickly done with DEL-key) and then request immediate synchronisation from component's elements. Unused surfaces are not removed automatically from the list!

Standardised surface heat transfer coefficients

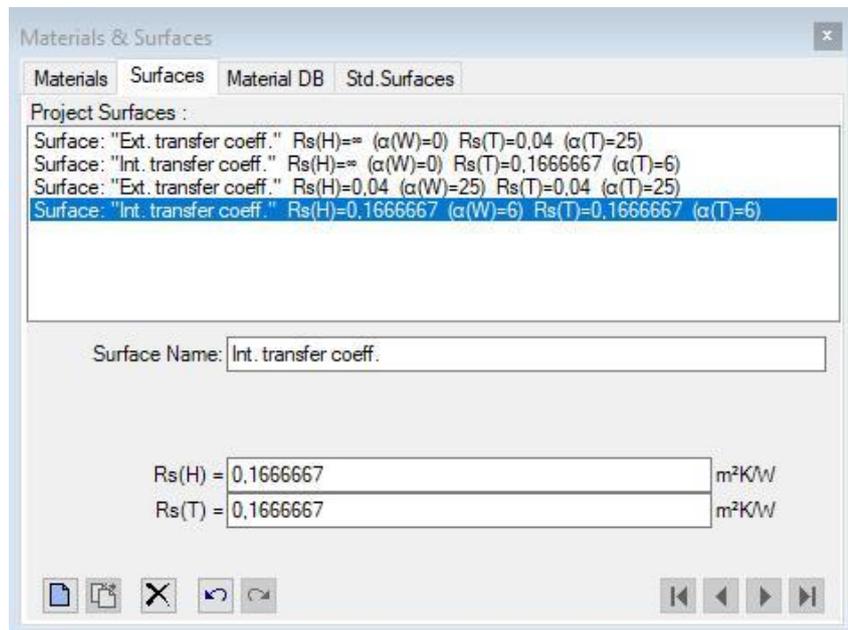
The tab "Std.Surfaces" provides a list of (read only) normative surface properties (standard values of R_{si} and R_{se}). You can find these [reference values within the description of U-Value calculator](#).

Important: While assessing heat loss different values must be used within the component model as those for the mould and condensation assessment! That means you will have to adjust respective R_{si} and R_{se} values depending on the assessment type.



See also: [Element Editor](#), [Elements 2D](#), [Element list](#), [Surface Editor window](#)

Surface Editor window



Surface Editor is used to maintain properties of surface. This window is displayed as part of [Surfaces window](#).

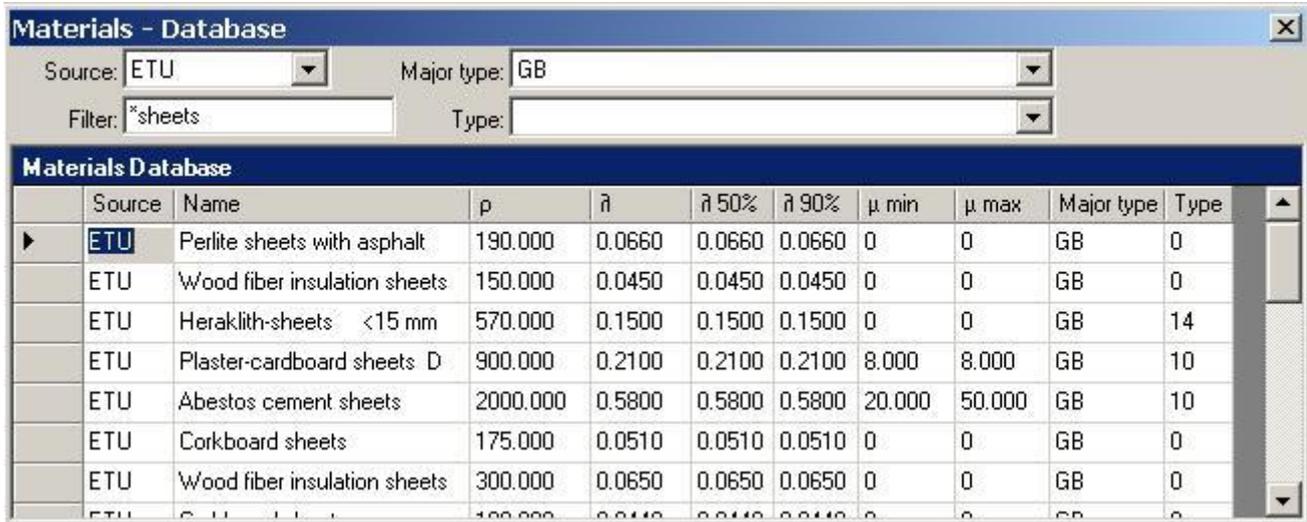
Surface name	Name of surface. It is used for documentation purpose.
RS (W) (T)	Defines the thermal <i>transfer resistance</i> of the surface. $R_s = 1 / \alpha$

Important: Changes to surface properties in the project specific [surface list](#) do not automatically propagate to properties of construction elements. Applying new surface properties to selected elements must be done explicitly - either by a double-click onto a list item or via a drag-drop operation from the [surface list](#).

Note: You shall switch between input fields by using the TAB-key. Using that key confirms the data entry. On the other hand it provides a good alternative compared to permanent mouse use.

Materials Database window

Material Database contains materials from DIN, ÖNORM, ETU, IBO, BPHDB.COM, Baubook/öbox, ArchiPHYSIK and ECOTECH. This window will be opened from the menu **Tools** → **Material Database** and from the menu **Tools** → **Materials Database**.



By selecting required filter criteria (Source, Major type, Type and Name filter) one can reduce amount of shown materials to those currently searched for. The filter setting is retained between application executions (will be saved together with application settings).

The context menu provides a command to reset the filter criteria (Source, Major type, Type and Name filter) thus showing all available data without any filtering.



A double-click onto a data selector (left to data fields) will assign material properties to currently selected material element(s) of the model. Such assignment can be executed by a *drag-drop with the right mouse button* also - see below.

Source	Provides the selection of shown materials by their data source. If empty entry is selected data from all sources is shown.
Filter Name-Filter	Provides the selection of shown materials by a substring contained in their name(e.g. Brick).
Major Type	Provides the selection of shown materials by their major type. If empty entry is selected data of all major types is shown. Remark: List of available major types is source specific.
Type	Provides the selection of shown materials by their type. If empty entry is selected data of all types is shown. Remark: List of available types is source specific.
Material list	Displays a list of materials in the database, possibly filtered according to selection criteria specified by <i>Source</i> , <i>Name-Filter</i> , <i>Major type</i> and <i>Type</i> .
Double-click left mouse button out of data record mark	Transfer material properties of the selected material onto currently selected element(s) within element list
Dragging with left mouse button out of data record mark	Transfer material properties of the material dragged with the mouse onto the element being currently edited in element editor or: Transfer material properties of the material dragged with the mouse onto the element(s) currently selected within element list or: Insert the material dragged with the mouse into the project own material list
lick left mouse button	Selection of a material

Remark: If there are several elements selected (from [element list](#)) then a double-click onto the material line (or drag-drop of material line) result in the assignment of material properties to all selected elements. Such bulk element assignment has to be confirmed by the user. If within selected elements there are some of incompatible type (non material) further user confirmation is requested.

Functional details summarized:

- Drag-Drop of material into [material window](#): Drag-Drop with right mouse button from record mark (left to data fields).
- Drag-Drop of material into [element editor](#): Drag-Drop with right mouse button from record mark (left to data fields).
Remark: A Drag-Drop of a material onto [element editor](#) is only possible if edited element is of sufficient type (material box).
- Drag-Drop of material onto [element list](#) or [Elemente2D window](#): Drag-Drop with right mouse button from record mark (left to data fields).
- *Double-click* onto a data record mark (left to data fields)

Remark: Drag-Drop is being initiated with right mouse button out of data record mark (left to data fields).

Remark: For technical reasons there are some columns hidden from the user. While tabbing between the fields the text cursor might hide also - simply press the TAB key again to reveal the focus.

Remark: Materials database data is stored within [the file "Baustoffe.xml"](#) within installation folder of the application and is loaded during this window is opened. You can modify this file within Excel or text editor if required.

Remark: If [Baubook/öbox-Data](#) or [BPHDB.COM-data](#) is available within the [Download-Folder](#) it will be loaded during this window is opened (the load operation might lead to slight delay before the window is displayed).

Materials -Database menu

While the [Materials Database window](#) is focused the main menu of the application is extended with the window specific menu "Materials DB":



Reset Filter	Reset the filter criteria (Source, Major type, Type and Name filter) thus showing all available data without any filtering.
Baubook(ÖBOX) Data	The menu specific for management of baubook(öbox) material data
Check for Updates...	A test for updated data against the Baubook/öbox server is initiated via internet connection
Download Updates...	A test for updated data against the Baubook/öbox server and a download (if new data is available) is initiated via internet connection
Include Baubook(ÖBOX) Data	Include baubook(öbox)-data (if available) within material database window . This, if turned off, allows you to exclude baubook(öbox)-data from the view even if the baubook(öbox) data file is available. See also: Application setting "Baubook(öbox) data included"
BPHDB Data	The menu specific for management of BPHDB.COM data
Check for Updates...	A test for updated data against the BPHDB.COM server is initiated via internet connection
Download Updates...	A test for updated data against the BPHDB.COM server and a download (if new data is available) is initiated via internet connection
Include BPHDB Data	Include BPHDB.COM-data (if available) within material database window . This, if turned off, allows you to exclude BPHDB.COM-data from the view even if the BPHDB data file is available. See also: Application setting "BPHDB.COM data included"

Remark: To execute an update check or the download of [Baubook/öbox](#) or [BPHDB.COM](#) data an internet connection is required.

Remark: Downloaded data will be saved to the [Download-Folder](#) specified within application settings.

See also: [Element Editor](#), [Materials window](#), [Baustoffkatalog ON V 31](#), [Material Database Baubook/ÖBOX](#), [Material Database BPHDB.COM](#), [Material Database ArchiPHYSIK \(APH\)](#), [Material Database ECOTECH](#)

Materials window

The Materials window is used to create and maintain the list of project specific material properties.

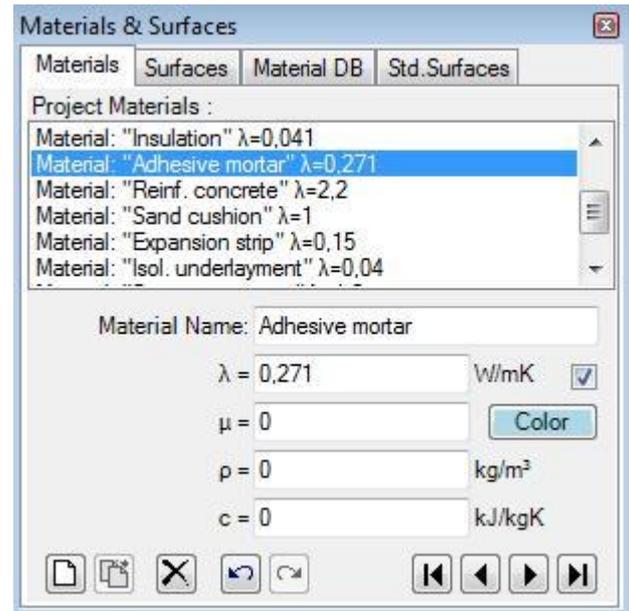
The list of materials offered in this window can be [synchronised](#) with materials directly entered and used in the model, thus all materials constituting the component are displayed here also. Materials can be [merged from other projects](#) and combined in this list.

The primary use of this window is to prepare lists of materials which, in cooperation with other windows, allow significant simplification and acceleration during the input of element properties for material boxes. Materials already prepared within another project can be [merged](#) to the list also.

Upper part of the window lists [materials](#). The bottom part – [Material Editor](#) - allows definition of material properties.

Important: Changes to material properties in the project specific materials list do not automatically propagate to properties of construction elements. Applying new material properties to selected elements must be done explicitly - either by a double-click onto a list item, via a drag-drop operation from the material list or by [the bulk replacement of properties](#).

Remark: Additional materials can be also selected from [Material Database window](#) by *drag-drop* from database ([Material Database](#) can be opened from the Tab "Material DB"). Dropping some material identical to the one in the list will be inhibited.



Material list

A double-click onto a line representing material in this window results in an assignment of material properties of selected material to currently selected material elements (selected from [elements list](#)).

Also, by *drag-drop operation* (by dragging with the mouse), a material line can be dragged onto [element editor](#) window resulting in assignment of material properties to the element currently shown there.

Materials window can be opened either from [main menu](#) *View*→*Data Input & Entry*→*Materials* or from [element editor](#) by a *double-click onto material name, conductivity and other material property input fields*.

Double-click left mouse button	Transfer of material properties of selected material onto currently selected element(s) in the element list .
Dragging with left mouse button	Transfer of material properties of dragged material to material element currently edited in element editor , or: Transfer of material properties of dragged material to currently selected material element(s) in the element list .
Click right mouse button	Exposes the context menu of material editing functions .
Click left mouse button	Selection of material from material list. Properties of selected material are displayed in Material Editor for further processing.

Remark: If there are several elements selected (from element list) then a double-click onto the material line (or drag-drop of material line) result in the assignment of material properties to all selected elements. Such bulk element assignment has to be confirmed by the user. If within selected elements there are some of incompatible type (non material) further user confirmation is requested. Error message is shown if neither element matches the expected type.

 New	Appends a new material onto the end of the material list
 Delete	Deletes the currently selected material from the list. Remark: The material is deleted from the material list only.
 Undo(Ctrl-Z)	Undoes (reverts) the latest action(s) in the given editing context .
 Redo(Ctrl-Y)	Redoes (reapplies) the action undone in the given editing context .
	Selects First, Previous, Next or Last material.

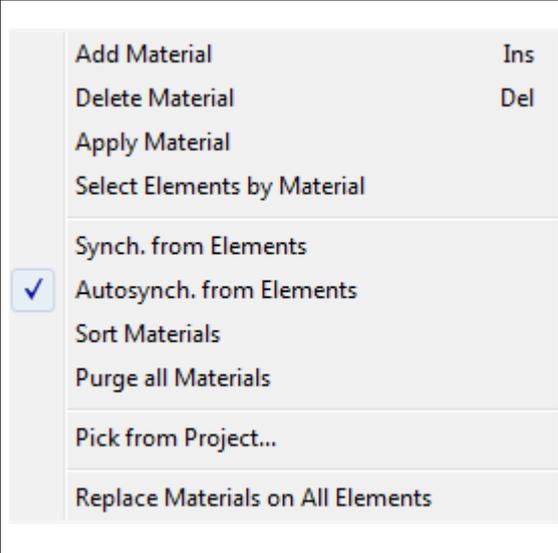
Material selection

A click onto material's line in the material list selects corresponding material for further processing.

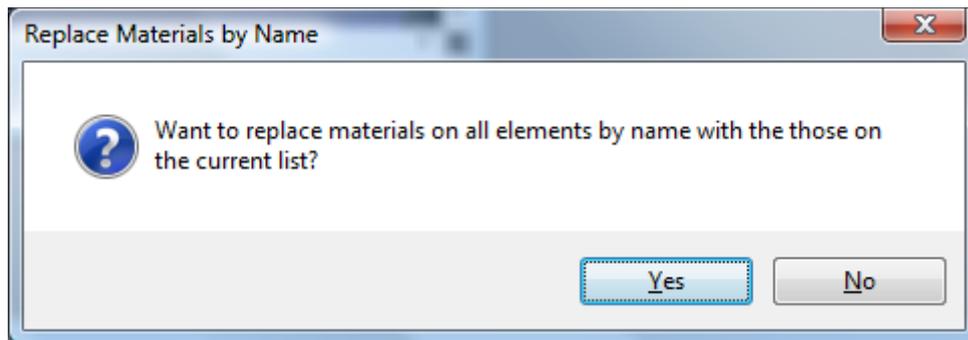


Selection can be done by arrow keys or by clicking the buttons Next/Prev. also.

Functions of the context menu

	
Add Material INS-Key	A new material is appended at the end of the list.
Delete Material DEL-Key	Deletes selected material from the list.
Apply Material	Transfer material properties of a selected material onto currently selected element(s) in the element list . Remark: Visible when exactly one surface is selected.
Select Elements by Material	Selects all elements in the element list having one specific material assigned. Remark: Visible when exactly one surface is selected. Remark: For 3D-Layered models the selection operation applies only to currently active layer.
Synch. from Elements	Executes immediate synchronisation of material list from properties of all elements currently in the modelled component
Autosynch. from Elements	The synchronisation of material list from properties of all elements in modelled component will be executed automatically (checkmark is visible) and can be turned on or off here.
Sort Materials	Materials will be sorted by <i>name, λ, μ, ρ</i> and <i>c</i> .
Purge all Materials	Removes all materials from the list
Pick from Project...	Materials saved within another project will be read and appended to the list.

	<u>Remark</u> : After importing the data from another project the (optional) bulk replacement of material properties on all elements with those just imported will be offered.
Replace Materials on All Elements	The material properties of all elements will be replaced by name with those from the current material list (bulk replacement). <u>Remark</u> : Shall duplicate name appear in the list only the first instance in the order will be applied. <u>Remark</u> : For 3D-Layered models the operation applies only to currently active layer.



Synchronisation of material list

The list of materials offered in this window can be synchronized (extended) with materials currently in use in the modelled component, thus all materials used in the model are also displayed. This provides the ability to transfer properties directly entered in [element editor](#) to the list and have them at hand as additional entries.

Synchronisation can be explicitly requested from the [context menu](#) of the material list ("Synch. from Elements") or will automatically be executed on each [element selection change](#) when the [application setting "Autosynch. from Elements"](#) is active.

Synchronisation executes by following steps:

- All elements of modelled component are analysed.
- Material properties of each material element are extracted.
- If no such material (name, lambda, my) can be found in material list it is appended to it.
- Materials appended during this synchronisation run and appended at the end of the list are sorted alphabetically.

This procedure assures, that entries are retained in the list and only "missing" materials will extend it.

Remark: Automatic synchronisation of list contents, if active, occurs on each [element selection change](#). Changes to element properties (in [Element Editor](#)) alone do not result in automatic synchronisation yet. Immediate synchronisation can be explicitly requested from the [context menu](#) of the material list ("Synch. from Elements").

Tip: To have the list showing only materials actually used in the modelled component delete all materials from the list (this can be quickly done with DEL-key) and then request immediate synchronisation from component's elements. Unused materials are not removed automatically from the list!

Remark: Even if values of ρ (density) and c (heat capacity) are transferred from material database, these values cannot be modified unless the [licence feature HARMONIC or TRANSIENT](#) is available (these data is not used during nor required for the steady state simulation).

See also: [Element Editor](#), [Elements 2D](#), [Element list](#), [Material Editor](#)

Material Editor window

Material Editor is used to maintain properties of material. This window is displayed as part of [materials window](#).

Material Name: Adhesive mortar

$\lambda = 0,271$ W/mK

$\mu = 0$

$\rho = 0$ kg/m³

$c = 0$ kJ/kgK

Material Name	Name of material. It is used for documentation purpose.
λ (lambda)	<i>Thermal Conductivity</i> of the material.
μ (mue)	<i>Water vapour diffusion resistance</i> of the material.
ρ (rho)	<i>Mass density</i> of the material.
c (ce)	<i>Heat capacity</i> of the material.
Colour and AutoColour	<p>Provides the ability to assign any distinctive colour to the element identifying it in graphical views and evaluations.</p> <p>To select the colour click onto the coloured rectangle labelled "Colour" - it will expose <i>colour selection dialog</i>.</p> <p>Remark: By default the colour is automatically assigned when change to lambda value is applied, based on predefined colour table.</p> <p>Removing the checkmark will turn the "auto-colour by lambda" application setting off thus assigned colour will be retained during changes to the value of lambda.</p>
New	Adds a new entry to the material list and selects it for editing.
Delete	Deletes the currently selected material from material list .
Prev.	Selects previous material in the material list for editing.
Next	Selects next material in the material list for editing.

Important: Changes to material properties in the project specific [material list](#) do not automatically propagate to properties of construction elements. Applying new material properties to selected elements must be done explicitly - either by a double-click onto a [material list](#) item or via a drag-drop operation from [material list](#).

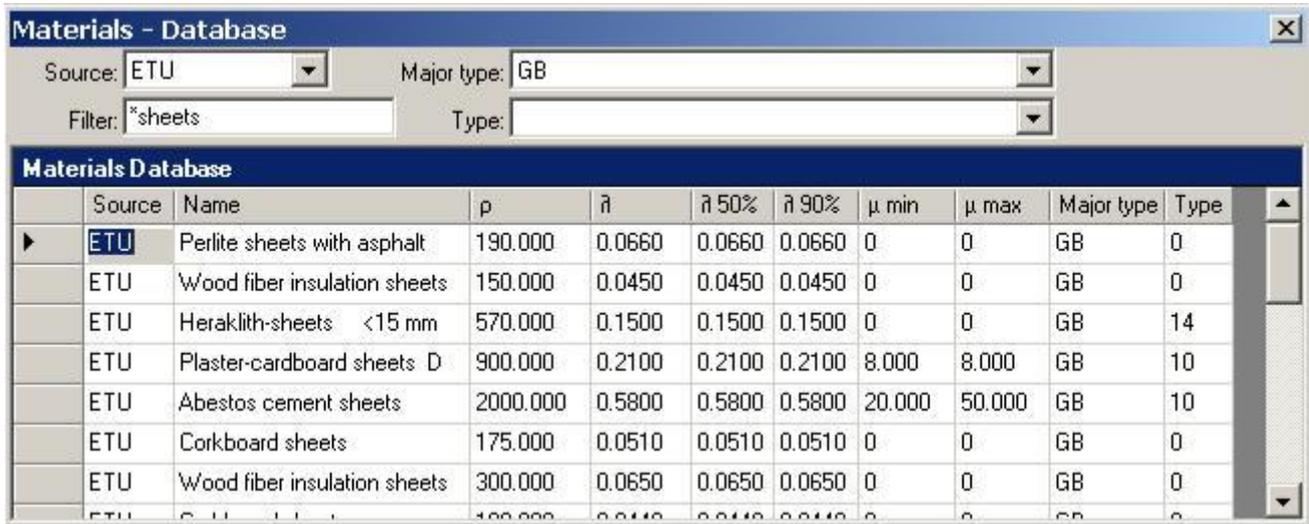
Remark: The [equivalent heat conductivity of small air cavities and air gaps](#) can be calculated with the [air cavity calculator tool](#).

Note: You shall switch between input fields by using the TAB-key. Using that key confirms the data entry. On the other hand it provides a good alternative compared to permanent mouse use.

See also: [Materials window](#), [Air Cavity Calculator \(Tool\)](#)

Materials Database window

Material Database contains materials from DIN, ÖNORM, ETU, IBO, BPHDB.COM, Baubook/öbox, ArchiPHYSIK and ECOTECH. This window will be opened from the menu **Tools**→**Material Database** and from the menu **Tools**→**Materials Database**. The main application's menu is extended with "**Materials-Database**" menu.



By selecting required filter criteria (Source, Major type, Type and Name filter) one can reduce amount of shown materials to those currently searched for. The filter setting is retained between application executions (will be saved together with application settings).

The context menu provides a command to reset the filter criteria (Source, Major type, Type and Name filter) thus showing all available data without any filtering.



A double-click onto a data selector (left to data fields) will assign material properties to currently selected material element(s) of the model. Such assignment can be executed by a *drag-drop with the right mouse button* also - see below.

Source	Provides the selection of shown materials by their data source. If empty entry is selected data from all sources is shown.
Filter Name-Filter	Provides the selection of shown materials by a substring contained in their name (e.g. Brick).
Major Type	Provides the selection of shown materials by their major type. If empty entry is selected data of all major types is shown. <u>Remark:</u> List of available major types is source specific.
Type	Provides the selection of shown materials by their type. If empty entry is selected data of all types is shown. <u>Remark:</u> List of available types is source specific.
Material list	Displays a list of materials in the database, possibly filtered according to selection criteria specified by <i>Source</i> , <i>Name-Filter</i> , <i>Major type</i> and <i>Type</i> .
Double-click left mouse button onto data record mark	Transfers material properties of the selected material onto currently selected element(s) within element list
Dragging with left mouse button out of data record mark	Transfer material properties of the material dragged with the mouse onto the element being currently edited in element editor or: Transfer material properties of the material dragged with the mouse onto the element(s) currently selected within element list

	or: Insert the material dragged with the mouse into the project own material list
lick left mouse button	Selection of a material

Remark: If there are several elements selected (from [element list](#)) then a double-click onto the material line (or drag-drop of material line) result in the assignment of material properties to all selected elements. Such bulk element assignment has to be confirmed by the user. If within selected elements there are some of incompatible type (non material) further user confirmation is requested.

Functional details summarized:

- Drag-Drop of material into [material window](#): Drag-Drop with right mouse button from record mark (left to data fields).
- Drag-Drop of material into [element editor](#): Drag-Drop with right mouse button from record mark (left to data fields).
Remark: A Drag-Drop of a material onto [element editor](#) is only possible if edited element is of sufficient type (material box).
- Drag-Drop of material onto [element list](#) or [Elemente2D window](#): Drag-Drop with right mouse button from record mark (left to data fields).
- *Double-click* onto a data record mark (left to data fields)

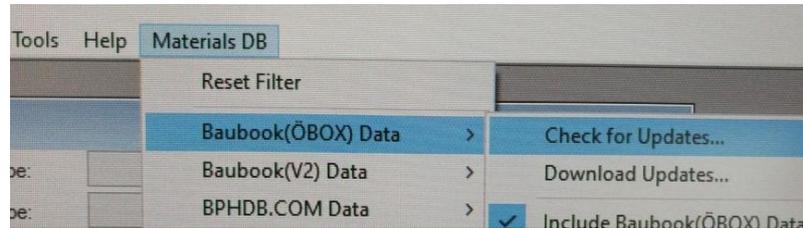
Remark: Drag-Drop is being initiated with right mouse button out of data record mark (left to data fields).

Remark: For technical reasons there are some columns hidden from the user. While tabbing between the fields the text cursor might hide also - simply press the TAB key again to reveal the focus.

Remark: Materials database data is stored within [the file "Baustoffe.xml"](#) within installation folder of the application and is loaded during this window is opened. You can modify this file within Excel or text editor if required.

Remark: If [Baubook/öbox-Data](#) or [BPHDB.COM-data](#) is available within the [Download-Folder](#) it will be loaded during this window is opened (the load operation might lead to slight delay before the window is displayed).

Materials- Database menu



Menu [Tools](#)→*Materials Database*. The main application's menu is extended with "[Materials-Database](#)" (Custom)menu.

While the [Materials Database window](#) is focused the main menu of the application is extended with the window specific menu "Materials DB":

Reset Filter	Reset the filter criteria (Source, Major type, Type and Name filter) thus showing all available data without any filtering.
Baubook(ÖBOX)Data	The menu specific for management of baubook(öbox) material data
Check for Updates...	A test for updated data against the Baubook/öbox server is initiated via internet connection.
Download Updates...	A test for updated data against the Baubook/öbox server and a download (if new data is available) is initiated via internet connection.
Include Baubook(ÖBOX)Data	Include baubook(öbox)-data (if available) within material database window . This, if turned off, allows you to exclude baubook(öbox)-data from the view even if the baubook(öbox) data file is available. See also: Application setting "Baubook(öbox) data included"
BPHDB Data	The menu specific for management of BPHDB.COM data .
Check for Updates..	A test for updated data against the BPHDB.COM server is initiated via internet connection.
Download Updates...	A test for updated data against the BPHDB.COM server and a download (if new data is available) is initiated via internet connection.
Include BPHDB Data	Include BPHDB.COM-data (if available) within material database window . This, if turned off, allows you to exclude BPHDB.COM-data from the view even if the BPHDB data file is available. See also: Application setting "BPHDB.COM data ncluded" .

Remark: To execute an update check or the download of [Baubook/öbox](#) or [BPHDB.COM](#) data an internet connection is required.

Remark: Downloaded data will be saved to the [Download-Folder specified within application settings](#).

See also: [Element Editor](#), [Materials window](#), [Baustoffkatalog ON V 31](#), [Material Database Baubook/ÖBOX](#), [Material Database BPHDB.COM](#), [Material Database ArchiPHYSIK \(APH\)](#), [Material Database ECOTECH](#)

Baustoffkatalog ON V 31

In der Anlehnung an Abschnitt 1 vom "Katalog für wärmeschutztechnische Rechenwerte von Baustoffen und Bauteilen", ON V 31 1. Auflage, Ausgabe 1. Dezember 2001, des Österreichischen Normungsinstitutes, Serie "Bauwesen" haben wir die typischen Baustoffe zusammengestellt.

Der Kataloginhalt soll im gesamten Bundesgebiet eine einheitliche Beurteilung von wärmeschutztechnischen Maßnahmen ermöglichen, wie sie laufend bei Planungen, bei den Durchführungen von Neubauten und in verstärktem Maße zukünftig auch bei Umbauten bzw. Adaptierungsmaßnahmen sowie im Betrieb auftreten; außerdem soll er als Grundlage für einfache Überprüfungen von an die Baustellen angelieferten Materialien dienen, um eine weitestgehende Übereinstimmung zwischen dem geplanten und dem zur Ausführung gelangenden Wärmeschutz zu gewährleisten.

...

Der ... Katalog stellt eine Weiterentwicklung der vom Bundesministerium für Bauten und Technik im Jahre 1979 für den staatlichen Hochbau herausgegebenen Richtlinien für maßgebliche Wärmeschutzrechenwerte dar. In der Neuauflage sind nun auch die europäischen Entwicklungen unter Bedachtnahme auf eine optimale Erfüllung der Umsetzung der Richtlinie des Rates vom 21. Dezember 1988 zur Angleichung der Rechts- und Verwaltungsvorschriften der Mitgliedsstaaten über Bauprodukte, Anhang I, Pkt. 6 – ENERGIEEINSPARUNG UND WÄRMESCHUTZ – bei der Planung und Errichtung aller Gebäude oder Räume zur Sicherstellung von deren Gebrauchstauglichkeit über einen jeweils aus wirtschaftlicher Sicht festzulegenden Zeitraum sowie die daraus abgeleiteten Normungsgrundlagen innerhalb von CEN/TC 89 "Wärmeschutz von Gebäuden und Bauteilen" sowie die ÖNORMEN-Serie B 8110 "Wärmeschutz im Hochbau" des österreichischen Spiegelgremiums – Fachnormenausschuss (FNA) 175 – berücksichtigt.

...

Die verschiedenen normativen Berechnungsverfahren für den Nachweis der Erfüllung der wärmeschutztechnischen Anforderungen und Kennwerte setzen die Verwendung zutreffender Rechenwerte für Baustoffe, Bauteilschichten, Hohlräume oder Bauteile voraus. Diese Werte können – sofern sie nicht in ÖNORMEN enthalten sind – dem nun vorliegenden "Katalog für empfohlene Wärmeschutz-Rechenwerte von Baustoffen und Baukonstruktionen" entnommen oder auch aus zertifizierten Produktkennwerten abgeleitet werden.

...

Der nun vorliegende völlig neu überarbeitete Katalog für empfohlene Wärmeschutzrechenwerte für Baustoffe und Baukonstruktionen ist ein grundlegendes Werk für dieses Fachgebiet und stellt für Architekten und Ingenieurkonsulenten, Zivilingenieure, Bauunternehmen, Baubehörden, Bausachverständige und Interessensvertretungen ein unverzichtbares Planungshilfsmittel dar.

...

Der Abschnitt 1 über die Wärmeleitfähigkeit von Bau- und Dämmstoffen enthält die Angaben über den zutreffenden Bereich der Rechenwerte (obere und untere Grenze) der Wärmeleitfähigkeit. Sofern die erreichbaren Grundlagendaten eine Bereichsangabe nicht zulassen, wird nur ein Wert für die Wärmeleitfähigkeit angeführt. Die Werte für die spezifische Wärmekapazität sind ebenfalls unter Berücksichtigung des baupraktischen Feuchtegehalts der Baustoffe angegeben.

...

Empfohlene Wärmeschutz-Rechenwerte für Baustoffe

Der Abschnitt 1 über die Wärmeleitfähigkeit von Baustoffen, einschließlich Dämmstoffen gibt für jedes in den Katalog aufgenommene Material die Dichte (DICHTE), die Wärmeleitfähigkeit (WÄRMELEITFÄHIGKEIT) für mittlere Bauverhältnisse ($U_{23/80}$) und die spezifische Wärme (SPEZ.W.) an. Die Rechenwerte der Wärmeleitfähigkeit in $W/(m \cdot K)$ werden als obere Grenzwerte (90%-Fraktile) und so weit dies möglich auch als Mittelwerte (50%-Fraktile) angegeben, die von 90% bzw. 50% der gemessenen oder überwachten Bauprodukte des betreffenden Materialtyps unterschritten werden.

Die oberen Grenzwerte sind zur Berechnung der U-Werte und der thermischen Leitwerte von Bauteilen heranzuziehen. Die Differenz zwischen dem oberen Grenzwert und dem Mittelwert gibt Aufschluss über die Breite der Verteilung der Messwerte (der Wärmeleitfähigkeit eines Baustoffes im betreffenden Dichtebereich) und ermöglicht die Abschätzung des unteren Grenzwertes der Wärmeleitfähigkeit.

Die unteren Grenzwerte sind zur Berechnung von Oberflächentemperaturen in Bauteilen heranzuziehen, sofern dies die gegebenen Randbedingungen oder das Auftreten von Wärmequellen im Bauteil erfordern.

Diese Form der Angabe der Wärmeleitfähigkeit als oberen Grenzwert und Mittelwert entspricht den inzwischen gefassten Beschlüssen in den europäischen Normungsgremien. Sie setzt allerdings voraus, dass für jeden der angegebenen Baustoffe oder Bauprodukte genügend Messwerte über deren Wärmeleitfähigkeit zur Verfügung stehen, um eine statistische Auswertung zu rechtfertigen. Diese Voraussetzung ist für die Wärmedämmstoffe in der Regel erfüllt, für andere konventionelle Baustoffe nur teilweise erfüllbar, für die Gruppe der Metalle und sonstiger Werkstoffe aber noch nicht gegeben.

Die statistische Auswertung stützt sich, analog dem Verfahren in CEN, ausschließlich auf Originalmesswerte, die im Rahmen von Zertifizierungsverfahren oder Güteüberwachungen (third party control) ermittelt wurden. Dabei werden nach der Berechnung der bestpassenden mittleren Regressionskurve die unteren und oberen Grenzwerte für die dichteabhängige Wärmeleitfähigkeit der betrachteten Baustoffart bestimmt, die von mindestens 90% der erfassten Daten erfüllt werden. Diese statistisch abgesicherten λ -Werte sind im Katalog daran erkennbar, daß dafür obere Grenzwerte und Mittelwerte für die dichteabhängigen Rechenwerte angegeben werden.

Für Baustoffe, für welche die erforderlichen Messwerte zur vollständigen statistischen Auswertung (noch) nicht zugänglich sind, werden nur die oberen Grenzwerte (90%-Fraktile) angeführt. Für Baustoffe, für die keine aktuellen Messwerte (oder nur Einzelwerte) der Wärmeleitfähigkeit existieren, werden in den Tabellen die bisherigen Rechenwerte beibehalten. Diese Werte sind in die Spalte für obere Grenzwerte eingetragen.

Für Werkstoffe oder Materialien, für die bisher keine Rechenwerte angegeben waren und für die bereits CEN-Werte existieren, wurden entsprechende Ergänzungen vorgenommen. Diese Ergänzungen betreffen insbesondere Stoffgruppen wie Metalle, Natursteine, Bodenmaterial und Kunststoffe.

Bei den angegebenen Dichtewerten ist im gegebenen Fall zwischen der Stoffrohndichte (z.B. Scherbenrohndichte eines Ziegelsteines) und der Komponentenrohndichte (z.B. eines Hochlochmauersteines oder eines Mauerwerks) zu unterscheiden. Die λ -Rechenwerte für das Vollziegelmauerwerk werden beispielsweise aus praktischen Gründen in Abhängigkeit von der Scherbenrohndichte des Ziegelsteines angegeben.

Für Baustoffe oder Baukomponenten, für die eine Interpolation der λ -Rechenwerte zwischen den einzelnen Dichtestufen zulässig ist, wird dies durch je ein Sternchen * vor und nach der Spaltenbezeichnung DICHTe angemerkt.

Die Rechenwerte für die spezifische Wärme sind zur Berechnung nicht stationärer Wärmeleitungsvorgänge in Bauteilen oder im Boden und zur Abschätzung des Wärmespeichervermögens von Bauteilen erforderlich.

Diese angegebenen Rechenwerte stammen aus dem Katalog für empfohlene Wärmeschutz-Rechenwerte 1979, sofern sie nicht durch Rechenwerte aus der ÖNORM EN ISO 10456 ersetzt wurden. Eine bessere Absicherung dieser Rechenwerte durch neuere Messungen ist derzeit nicht möglich.

Die spezifische Wärme von Baustoffen wird in den CEN-Dokumenten für den Trockenzustand und für 10 °C in J/(kg·K) angegeben. Für die nationalen Tabellenwerte, die für baupraktische Feuchtezustände zutreffen, wurde daher die erforderliche Umrechnung vorgenommen. ...

Für die gegenständliche Zusammenstellung der Rechenwerte wird die bisher verwendete Dimension kJ/(kg·K) beibehalten. Für Berechnungen von Wärmeströmen durch den Boden wird in CEN-Dokumenten auch die Dimension J/(m³·K) verwendet.

Im Katalog werden der Baustoff- oder der Materialbezeichnung (Materialtypen) in der Regel Ordnungsnummern vorgesetzt. Jeder Datenzeile (Baustoff oder Material bestimmter Dichte) ist eine Katalog-Nummer vorangestellt.

Overview about the building material- and material types.

1 MAURERWORK

- 1.102 ZIEGELMATERIAL (gem. ÖNORM B 3200; Reinders)
- 1.104 VOLLZIEGELMAUERWERK
- 1.106 HOCHLOCHZIEGELMAUERWERK (Altbestand)
- 1.108 HOCHLOCHZIEGELMAUERWERK (Neubau)
- 1.110 LANGLOCHZIEGELMAUERWERK
- 1.112 BETONHOHLSTEINMAUERWERK
- 1.114 PORENBETON
- 1.116 GIPSWANDBAUPLATTEN
- 1.118 ZWISCHENWANDZIEGEL

2 MASONRY AND PLASTER MORTAR

- 2.202 NORMALMAUERMÖRTEL
- 2.204 MAUERMÖRTEL MIT SONSTIGEN ZUSCHLÄGEN
- 2.206 MAUERMÖRTEL MIT BLÄHTON-ZUSCHLAG
- 2.208 MAUERMÖRTEL MIT PERLITE-ZUSCHLAG
- 2.210 PUTZMÖRTEL – AUSSENPUTZE
- 2.212 PUTZMÖRTEL – INNENPUTZE
- 2.214 PUTZMÖRTEL MIT SONSTIGEM LEICHTZUSCHLAG
- 2.216 PUTZMÖRTEL MIT PERLITE-ZUSCHLAG
- 2.218 PUTZMÖRTEL MIT EPS-ZUSCHLAG
- 2.220 SANIERPUTZMÖRTEL

3 CONCRETES AND SCREEDS

- 3.302 BETON MIT ZUSCHLÄGEN AUS NATÜRLICHEM GESTEIN
- 3.304 STAHLBETON
- 3.306 BETON MIT BLÄHTON-ZUSCHLAG
- 3.308 BETON MIT BLÄHSCHIEFER-ZUSCHLAG
- 3.310 BIMSBETON OHNE NATURSAND
- 3.312 HÜTTENBIMSBETON OHNE NATURSAND
- 3.314 BETON MIT LEICHTSPLITT-ZUSCHLAG
- 3.316 BETON MIT ZIEGELSPLITT-ZUSCHLAG
- 3.318 PORENBETON
- 3.320 BETON MIT EPS-ZUSCHLAG
- 3.322 HOLZSPANBETON
- 3.324 NICHTZEMENTGEBUNDENE ESTRICHBAUSTOFFE
- 3.326 ZEMENTGEBUNDENE ESTRICHBAUSTOFFE
- 3.328 KALKSANDSTEINMATERIAL

4 STANDARDIZED INSULATION MATERIALS

- 4.402 HOLZWOLLE-DÄMMPLATTEN (OHNE PORENVERSCHLUSS) W
- 4.404 HOLZWOLLE-DÄMMPLATTEN (MIT PORENVERSCHLUSS) WW-PV 4.406
- HOLZWOLLE-MEHRSCICHT-DÄMMPLATTEN (gem. ÖNORM B 6021) 4.408
- HOLZSPAN-DÄMMPLATTEN WS
- 4.410 DÄMMKORK (reinexpandiert; gem. ÖNORM B 6031)
- 4.412 DÄMMKORK (gepresst)
- 4.414 GEBUNDENE MINERALWOLLE (Glaswolle; gem. ÖNORM B 6035; Wärmestrom senkrecht zur Faser) 4.416
- GEBUNDENE MINERALWOLLE (Glaswolle; gem. ÖNORM B 6035; Wärmestrom parallel zur Faser) 4.418
- GEBUNDENE MINERALWOLLE (Glaswolle; gem. ÖNORM B 6035; gekrept)
- 4.420 GEBUNDENE MINERALWOLLE (Steinwolle; gem. ÖNORM B 6035; Wärmestrom senkrecht zur Faser) 4.422
- GEBUNDENE MINERALWOLLE (Steinwolle; gem. ÖNORM B 6035; Wärmestrom parallel zur Faser) 4.424
- GEBUNDENE MINERALWOLLE (Steinwolle; gem. ÖNORM B 6035; gekrept)
- 4.426 EXPANDIERTER POLYSTYROL PARTIKELSCHAUMSTOFF
(gem. ÖNORM B 6050; GPH-Nennwerte)
- 4.428 POLYSTYROL-EXTRUDERSCHAUMSTOFF XPS (Altbestand; gem. ÖNORM B 6053; Zellgas FCKW) 4.430
- POLYSTYROL-EXTRUDERSCHAUMSTOFF XPS (gem. ÖNORM B 6053; Zellgas HFCKW)
- 4.432 POLYSTYROL-EXTRUDERSCHAUMSTOFF XPS (gem. ÖNORM B 6053; Zellgas HFKW) 4.434
- POLYSTYROL-EXTRUDERSCHAUMSTOFF XPS (gem. ÖNORM B 6053; Zellglas Luft) 4.436
- POLYURETHAN-HARTSCHAUMSTOFF PUR (CFC-extrudiert, überwacht)
- 4.438 POLYURETHAN-HARTSCHAUMSTOFF PUR (CFC-extrudiert, nicht überwacht)
- 4.440 POLYURETHAN-HARTSCHAUMSTOFF PUR (Pentan-extrudiert, zw. diffusionsdichten Schichten) 4.442
- POLYURETHAN-HARTSCHAUMSTOFF PUR (Pentan-extrudiert, diffusionsoffen)
- 4.444 SCHAUMGLAS
- 4.446 HOLZFASERDÄMMPLATTEN (Nassverfahren)
- 4.448 HOLZWOLLE (gebunden mit Wasserglas)

5 WOOD AND WOOD-BASED MATERIALS

- 5.502 HOLZ UND SPERRHOLZ
- 5.504 HOLZSPANPLATTEN
- 5.506 HOLZSPANPLATTEN (ZEMENTGEBUNDEN)
- 5.508 HOLZSPANPLATTEN (GIPSGEBUNDEN)
- 5.510 OSB-PLATTEN
- 5.512 HOLZFASERPLATTEN (Nassverfahren)
- 5.514 MDF-PLATTEN (Trockenverfahren)

6 FILLS

- 6.602 SCHÜTTUNGEN AUS BLÄHGLIMMER (LOSE)
- 6.604 SCHÜTTUNGEN AUS KORKSCHROT (EXPANDIERT)
- 6.606 SCHÜTTUNGEN AUS SONSTIGEN PORIGEN STOFFEN
- 6.608 SCHÜTTUNGEN AUS SAND, KIES, SPLITT

7 FLOOR COVERINGS OF PLASTIC, RUBBER AND SIMILAR

- 7.702 LINOLEUM UND KORK
- 7.704 KUNSTSTOFF- UND GUMMIBELÄGE
- 7.706 TEXTILE BELÄGE

7.708 UNTERLAGE UND RÜCKENBESCHICHTUNG VON BODENBELÄGEN

8 INDIVIDUAL BUILDING MATERIALS

8.802 METALLE

8.804 GLAS, FLIESEN, KERAMIKVERKLEIDUNGEN

8.806 PLATTEN

8.808 THERMAX-PLATTEN (AUS VERMICULIT; GEBUNDEN)

8.810 PLATTEN AUS PERLITE UND FASERMATERIAL MIT ASPHALTEMULSION

8.812 PLATTEN AUS PERLITE, BLÄHTON UND ÄHNLICHEM MIT KUNSTHARZBINDUNG

8.814 PLATTEN AUS PERLITE

8.816 ABDICHTSTOFFE

8.818 KUNSTSTOFFMATERIAL

8.820 KAUTSCHUK

8.822 NATURSTEINE

8.824 BODENMATERIAL

8.826 DACHZIEGEL, DACHSTEINE

8.828 GASE

8.830 WASSER

8.832 DÄMMSTOFFE IN KONTAKT MIT DEM BODEN (FROSTSCHUTZ)

8.834 DICHTSTOFFE, DICHTBÄNDER UND WÄRMEBRÜCKENUNTERBRECHER

9 OTHER INSULATING MATERIALS

9.902 PIR

9.904 PHENOLHARZSCHAUM

9.906 HARNSTOFFSCHAUM

9.908 POLYESTERFASER

9.910 KORKSTEINPLATTEN

9.912 DÄMMKORK (PECHIMPRÄGNIERT)

9.914 KOKOSFASERMATTEN

9.916 BAUMWOLLE-DÄMMATTEN

9.918 SCHAFWOLLE

9.920 SCHILFBAUPLATTEN

9.922 ZELLULOSE (LOSE)

9.924 FLACHS

9.926 GLUED EPS-BOARDS

Material Database Baubook (Reference Data)

The "baubook" (earlier "öbox") (www.baubook.info/zentrale/) provides characteristic values of popular materials. The *baubook* data contains set of characteristic physical and ecological indicators. The dataset has been created in cooperation with "Österreichisches Institut für Baubiologie und -ökologie (IBO)". Data can be freely downloaded and used in calculations.

Registered users of baubook-plattform are automatically alerted by e-Mail-Newsletter if changes or extensions apply.

If you miss data for specific materials please contact Baubook, Energieinstitut Vorarlberg .

The reference and product indicators are based on Austrian standards and regulations and are provided for calculation of energy and ecology indicators created to fulfil formal requirements (energy pass). In general this data can be used in other countries also.

These data is only partially applicable to expertise opinions - especially regarding vapour diffusion calculations.

Die baubook GmbH übernimmt keinerlei Haftung für die bereitgestellten Richt- und Produktkennwerte. Es handelt sich um eine Aufzählung erhaltener und gesammelter Informationen. Die Datenbank erhebt daher keinen Anspruch auf Vollständigkeit und Richtigkeit. Die baubook GmbH haftet auch nicht für den Fall, dass eine Förderung aufgrund falscher Herstellerangaben nicht erteilt oder widerrufen wird.

Importing updated data.

Thanks to an agreement between vendors the data of the baubook-catalogue shall be used within AnTherm. While the [materials database window](#) is active you will find a [main menu item specific to baubook-functions](#) allowing you to initiate the check and download of updates from the baubook Server via internet.

Remark: To execute an update check and/or the download of baubook data an internet connection is required.

Remark: The data set of baubook data is stored within a [user specific folder \(as defined within application settings\)](#)

Material Database BPHDB.COM (Building Physics Database)

The Building Physics Database (www.bphdb.com) provides a place to declare and maintain data for building physics software.

The BPHDB.COM is Copyright (c) Kurt Battisti, Eichendorffstr. 1, 8010 Graz, Österreich.

Importing updated data.

Thanks to an agreement between vendors the data of the BPHDB.COM-catalogue shall be used within AnTherm. While the [materials database window](#) is active you will find a [main menu item specific to BPHDB.COM-functions](#) allowing you to initiate the check and download of updates from the [BPHDB.COM](#) Server via internet.

Remark: To execute an update check and/or the download of [BPHDB.COM](#) data an internet connection is required.

Material Database APH (ArchiPHYSIK Database)

The ArchiPHYSIK Database is Copyright (c) A-NULL Bauphysik GmbH, Österreich.

Thanks to an agreement between vendors the data of the ArchiPHYSIK-catalogue shall be used within AnTherm.

Remark: The data set "APH9_052011" reflects the state as it was friendly provided for AnTherm on 9th May 2011 by A-NULL Bauphysik GmbH, Kurt Battisti, and cannot be changed by the user.

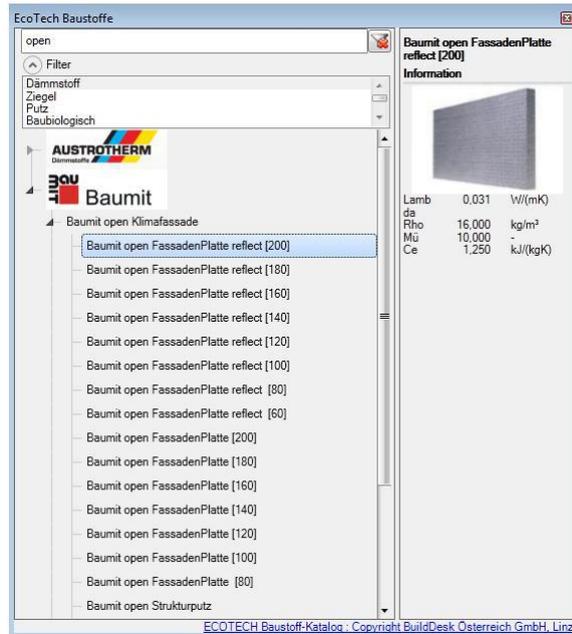
Remark: The data set is stored within a [user specific folder \(as defined within application settings\)](#)

Material Database Ecotech (ECOTECH-Catalogue)

The [ECOTECH Database](#) is Copyright (c) BuildDesk Österreich GmbH, Linz.

Thanks to an agreement between vendors the data of the ECOTECH-catalogue shall be used within AnTherm.

Remark: The data set "ecotech_062011" reflects the state as it was friendly provided for AnTherm on 15th June 2011 by BuildDesk Österreich GmbH, Mag. Hermann Jahrman, and cannot be changed by the user.



The ECOTECH Material Catalogue window uses the data of the ECOTECH Building Calculator and provides this data within AnTherm.

Remark: The ECOTECH Material Catalogue requires the "ECOTECH Building Calculator" to be installed already. If this installation is omitted, the functions are hidden (out of the [Main-menu](#) too).

On the left there is ECOTECH Material Catalogue Data displayed.

Respective materials are grouped together (e.g. standards, manufacturer) and further subdivided into subgroups.

The nodes of such a tree can be expanded by a click onto a triangle symbol or by a double click onto the name; further double click (or click onto the triangle symbol) closes that node.

The right part displays physical material properties (thermal conductivity λ , density ρ , heat capacity c , ...) and (optionally) images with further information (e.g. links to manufacturer information).

To use materials out of the catalogue do the following:

- Double-Click onto the material assigns that material to [the currently selected element\(s\)](#).
- A Drag&Drop of a material onto [Element Editor](#), [Elements 2D](#), [Elements 23](#) or [Element Browser](#) assigns that material to [the currently selected element\(s\)](#); when dropped onto [Materials window](#) appends it to the list.

Compare to: [Materials Database window](#)

Searching and filtering of materials

Materials can be filtered by name by entering some text into the field at the top of the window. There is no wildcard support! Whitespace separated filter segments will be filter like the continuous sequence. Entering "open" for example will show only materials of which names contain the word "open".

In addition to "name filter" one can filter materials by their auxiliary properties (e.g. "insulation", "brick"). These properties are defined by the manufacturer of the catalogue. Click onto (^) to show the list of selectable auxiliary filter properties.

Click onto "filter delete" symbol resets the filter and all material are shown again.

See also: [Element Editor](#), [Materials window](#), [Materials Database window](#), [Material Database ECOTECH](#)

Surfaces window

The Surfaces window is used to create and maintain the list of project specific surface properties.

The list of surfaces offered in this window can be [synchronised](#) with surfaces directly entered and used in the model, thus all surfaces constituting the component are displayed here also.

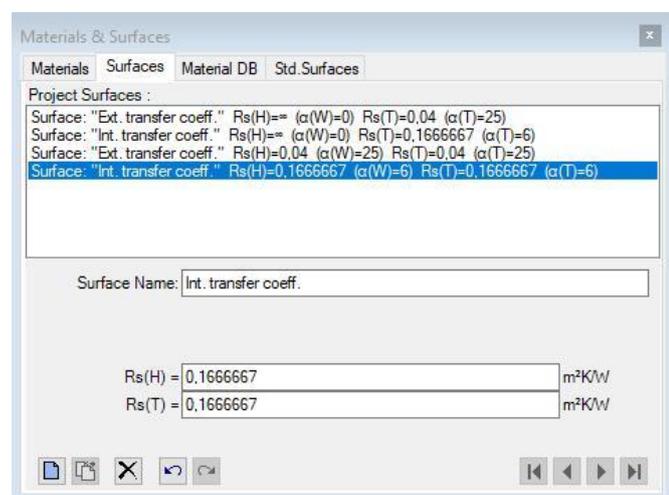
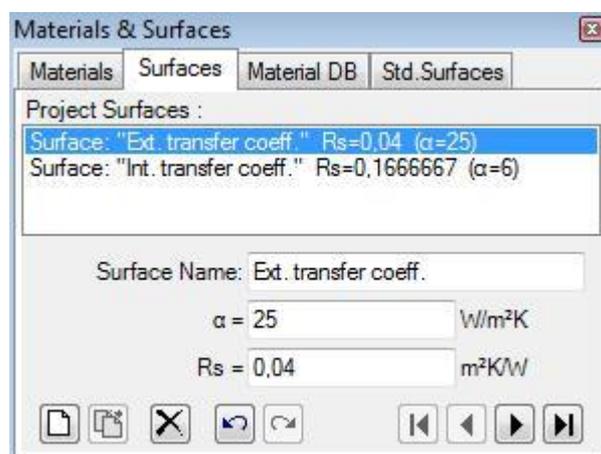
Surfaces can be [merged from other projects](#) and combined in this list.

The primary use of this window is to prepare lists of surfaces which, in cooperation with other windows, allow significant simplification and acceleration during the input of element properties for space boxes.

Upper part of the window [lists surfaces](#). The bottom part - [Surface Editor](#) - allows definition of surface properties.

Important: Changes to surface properties in the project specific surface list do not automatically propagate to properties of construction elements. Applying new surface properties to selected elements must be done explicitly - either by a double-click onto a list item, via a drag-drop operation from the surface list or by [the bulk replacement of properties](#).

Remark: The tab "[Std.Surfaces](#)" provides a list of (read only) [normative surface properties](#).



In AnTherm Version 10 you can change RS values for several room cells. Therefore, select the corresponding room cells in the "Element selection". Click the right mouse button, where you will find the function "change Rs values". Click again; it opens another window where you can change the RS values.

Surface list

A double-click onto a line representing surface in this window results in an assignment of surface properties of selected surface to currently selected space elements (selected from [elements list](#)).

Also, by *drag-drop operation* (by dragging with the mouse), a surface line can be dragged onto [element editor](#) window resulting in assignment of surface properties to the element currently shown there.

Surfaces window can be opened either from [main menu](#) *View→Data Input & Entry→Surfaces* or from [element editor](#) by a *double-click onto surface name, transfer coefficient or resistance input fields*.

Surface list	List of all surface properties entered
Double-click left mouse button	Transfer of surface properties of selected material onto currently selected elemen(s) in the element list .
Dragging with left mouse button	Transfer of surface properties of dragged surface to space element currently edited in element editor , or: Transfer of surface properties of dragged surface to currently selected space elemen(s) in the element list .
Click right mouse button	Exposes the context menu of surface editing functions .
Click left mouse button	Selection of surface from surface list. Properties of selected surface are displayed in Surface Editor for further processing.

Remark: If there are several elements selected (from [element list](#)) then a double-click onto the surface line (or drag-drop of surface line) result in the assignment of surface properties to all selected elements. Such bulk element assignment has to be confirmed by the user. If within selected elements there are some of incompatible type (non space) further user confirmation is requested. Error message is shown if neither element matches the expected type.

 New	Appends a new surface onto the end of the surface list
 Delete	Deletes the currently selected surface from the list. Remark: The surface is deleted from the surface list only.
 Undo(Ctrl-Z)	Undoes (reverts) the latest action(s) in the given editing context .
 Redo(Ctrl-Y)	Redoes (reapplies) the action lately undone in the given editing context .
	Selects First, Previous, Next or Last surface.

Surface selection

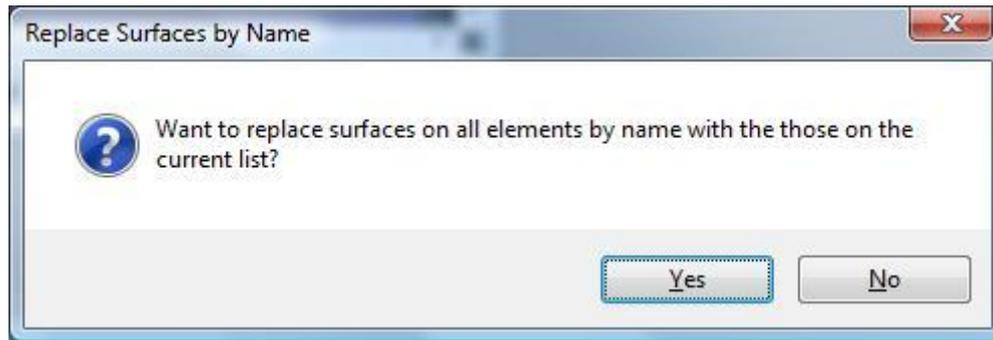
A click onto surface's line in the surface list selects corresponding surface for further processing.



Selection can be done by arrow keys or by clicking the buttons Next/Prev. also.

Functions of the context menu

Add Surface Ins Delete Surface Del Apply Surface Select Elements by Surface Synchronisation from Elements <input checked="" type="checkbox"/> Autosynchronisation from Elements Sort Surfaces Purge all Surfaces Pick from Project... Replace Surfaces on All Elements	
Add Surface INS-Key	A new surface is appended at the end of the list.
Delete Surface DEL-Key	Deletes selected surface from the list.
Apply Surface	Transfer surface properties of a selected surface onto currently selected element(s) in the element list . <u>Remark:</u> Visible when exactly one surface is selected.
Select Elements by Surface	Selects all elements in the element list having one specific surface assigned. <u>Remark:</u> Visible when exactly one surface is selected. <u>Remark:</u> For 3D-Layered models the selection operation applies only to currently active layer.
Synch. from Elements	Executes immediate synchronisation of surface list from properties of all elements currently in the modelled component
Autosynch. from Elements	The synchronisation of surface list from properties of all elements in the modelled component will be executed automatically (checkmark is visible) and can be turned on or off here.
Sort Surfaces	Surfaces will be sorted by name and alpha.
Purge all Surfaces	Removes all surfaces from the list
Pick from Project...	Surfaces saved within another project will be read and appended to the list. <u>Remark:</u> After importing the data from another project the (optional) bulk replacement of surface properties on all elements with those just imported will be offered.
Replace Surfaces on All Elements	The surface properties of all elements will be replaced by name with those from the current surface list (bulk replacement). <u>Remark:</u> Shall duplicate name appear in the list only the first instance in the order will be applied. <u>Remark:</u> For 3D-Layered models the operation applies only to currently active layer.



Synchronisation of surface list

The list of surfaces offered in this window can be synchronized (extended) with surfaces currently in use in the modelled component, thus all surfaces used in the model are also displayed. This provides the ability to transfer properties directly entered in [element editor](#) to the list and have them at hand as additional entries.

Synchronisation can be explicitly requested from the [context menu](#) of the material list ("Synch. from Elements") or will automatically be executed on each [element selection change](#) when the [application setting "Autosynch. from Elements"](#) is active.

Synchronisation executes by following steps:

- All elements of modelled component are analysed.
- Surface properties of each space element are extracted.
- If no such surface (name, α , R_s) can be found in surface list it is appended to it.
- Surfaces appended during this synchronisation run and appended at the end of the list are sorted alphabetically.

This procedure assures, that entries are retained in the list and only "missing" surfaces will extend it.

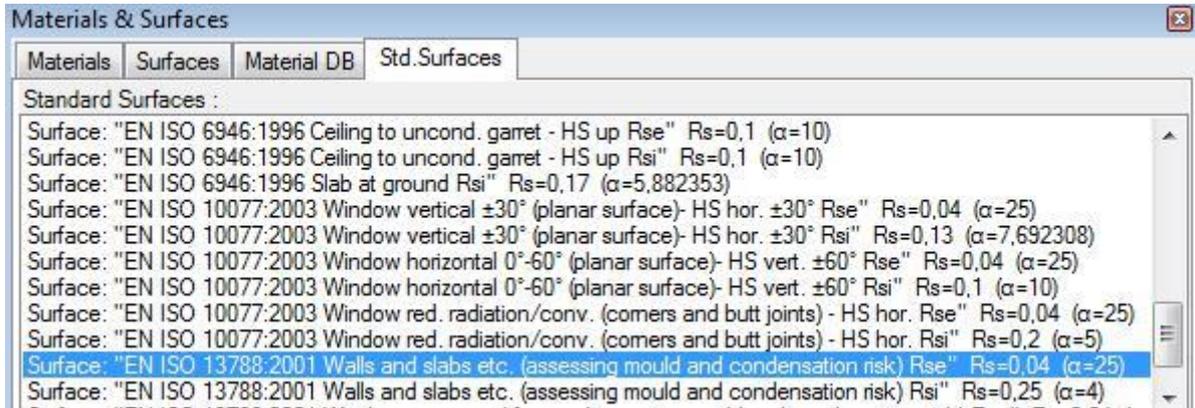
Remark: Automatic synchronisation of list contents, if active, occurs on each [element selection change](#). Changes to element properties (in [Element Editor](#)) alone do not result in automatic synchronisation yet. Immediate synchronisation can be explicitly requested from the [context menu](#) of the surface list ("Synch. from Elements").

Tip: To have the list showing only surfaces actually used in the modelled component delete all surfaces from the list (this can be quickly done with DEL-key) and then request immediate synchronisation from component's elements. Unused surfaces are not removed automatically from the list!

Standardised surface heat transfer coefficients

The tab "Std.Surfaces" provides a list of (read only) normative surface properties (standard values of R_{si} and R_{se}). You can find these [reference values within the description of U-Value calculator](#).

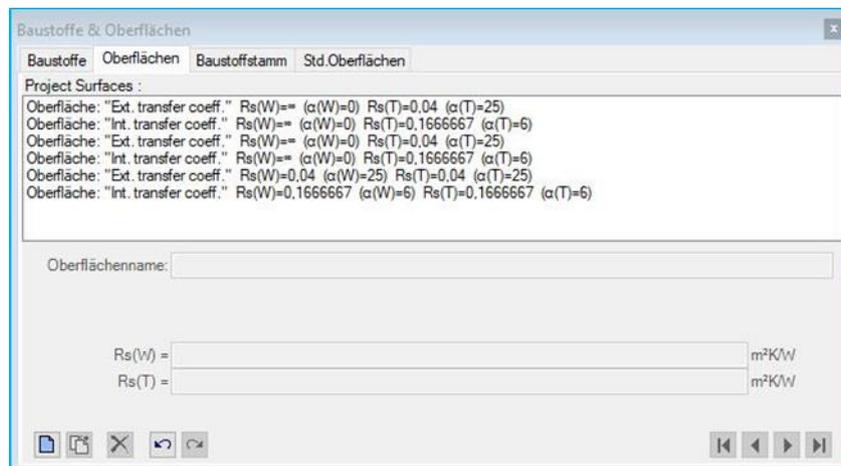
Important: While assessing heat loss different values must be used within the component model as those for the mould and condensation assessment! That means you will have to adjust respective R_{si} and R_{se} values depending on the assessment type.



See also: [Element Editor](#), [Elements 2D](#), [Element list](#), [Surface Editor window](#)

Surface Editor Window

Surface Editor is used to maintain properties of surface. This window is displayed as part of [Surfaces window](#).



Surface name	Name of surface. It is used for documentation purpose.
$R_S (H)$	Defines the thermal transfer resistance of the surface. $RS= 1 / \alpha$
$R_S (T)$	

Important: Changes to surface properties in the project specific [surface list](#) do not automatically propagate to properties of construction elements. Applying new surface properties to selected elements must be done explicitly - either by a double-click onto a list item or via a drag-drop operation from the [surface list](#).

Note: You shall switch between input fields by using the TAB-key. Using that key confirms the data entry. On the other hand it provides a good alternative compared to permanent mouse use.

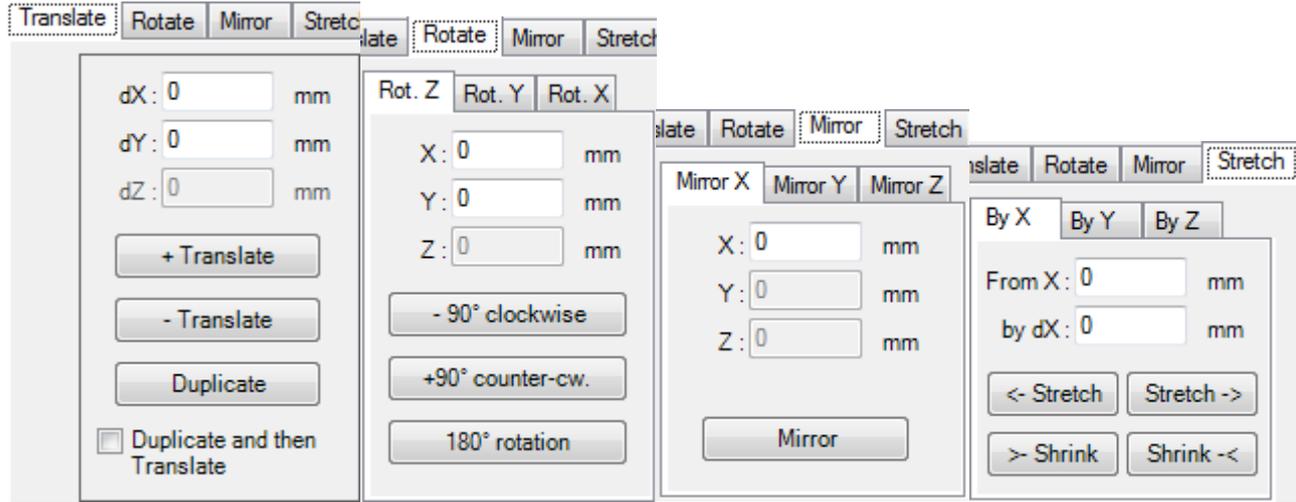
See also: [Surfaces window](#), [Alpha \(surface transmittance\) hidden \(application setting\)](#)

Transforms (Rotate, Mirror, Translate, Stretch, ...) window

The window Transforms combines operations of [Rotation](#), [Mirroring](#), [Translation](#) and [Stretching](#) applied to elements,

- [Translate window](#)
- [Rotate window](#)
- [Mirror window](#)
- [Stretch window](#)

The input of X, Y, Z coordinates of the point defining the transformation is common to both functions.



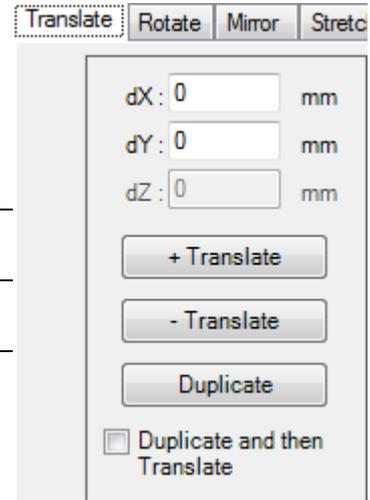
 	Undoes (reverts) the latest action(s) in the given editing context or Redoes (reapplies) the action lately undone in the given context .
Undo (Strg-Z) Redo (Strg-Y)	

See also: [Element Editor](#), [Context menu of element processing](#), [Element selection window](#)

Translate window

The Translate window allows translation of one or many [elements](#) (selected the [element list](#)) by specific distance (dX,dY,dZ).

The window Translate can be called from the [context menu of element processing](#) (a sub menu Translate).



+ Translate	Translates one or many elements (selected the element list) by specific vector (dX,dY,dZ)
- Translate	Translates one or many elements (selected the element list) by specific vector (-dX,-dY,-dZ)
Duplicate	Inserts a copy of currently selected element(s) into the element list . Remark: Holding SHIFT-key while that button is depressed will additionally swap coordinates of the duplicate. If several elements are selected their copies are appended to the end of element list retaining their order. After being inserted all these element copies are marked selected thus being ready for further processing. Remark: This function is identical to "Duplicate" in the context menu of

	element processing.
Duplicate and then Translate	If this checkbox is marked, "Duplicate" is executed prior to each "Translate". The translation is then applied to the copies (duplicates) of elements selected earlier. Remark: The functionality of "Duplicate and then Translate" provides a basis for the entry of complex components, especially those representing sloped constructions.
 Undo (Strg-Z) Redo (Strg-Y)	Undoes (reverts) the latest action(s) in the given editing context or Redoes (reapplies) the action lately undone in the given context .

The translation is applied equally to all elements selected from the [element list](#). If no elements are currently selected both buttons "+ Translate" and "- Translate" are inactive.

Remark: Repetitive combinations of "+ Translate" and "- Translate" can be used to revert translations.

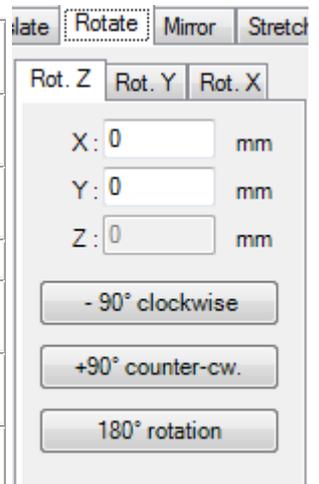
Remark: Z-coordinates of elements in 2D or Layered 3D projects are calculated from layer's thickness. Therefore the input field dZ is inactive in such projects.

See also: [Element Editor](#), [Context menu of element processing](#), [Element selection window](#), [Stretch window](#), [Rotate window](#), [Mirror window](#)

Rotate window

The window Rotate allows rotations of one or more [element\(s\)](#) selected from [element list](#). The window is opened from the [context menu of element processing](#) (submenu Rotate).

Rot. Z	Selects the rotation in the Z plane. The centre of rotation is defined by the input of X and Y coordinate values
Rot. Y	Selects the rotation in the Y plane. The centre of rotation is defined by the input of X and Z coordinate values
Rot. X	Selects the rotation in the X plane. The centre of rotation is defined by the input of Y and Z coordinate values
X, Y, Z	Coordinates of the rotation centre point
- 90° clockwise	All element(s) selected from element list are rotated around the rotation's centre.
+ 90° counter cw.	All element(s) selected from element list are rotated around the rotation's centre.
180° rotation	All element(s) selected from element list are rotated around the rotation's centre.
Rotate	Selection of the rotation function of this window
 Undo (Strg-Z) Redo (Strg-Y)	Undoes (reverts) the latest action(s) in the given editing context or Redoes (reapplies) the action lately undone in the given context .



The rotation is applied equally to all elements currently selected in the [element list](#). If there are no elements selected all *Rotation buttons* are set inactive.

Remark: Rotation in Y or X plane is available only in 3D projects.

Remark: Element rotation are done only in 90 degrees steps to keep the orthogonal structure of the input model.

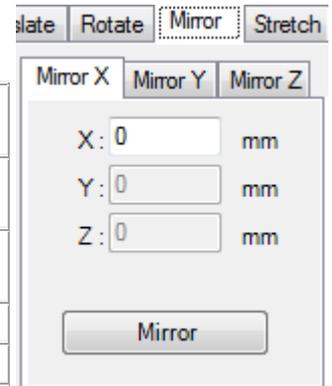
Remark. To rotate elements of belonging together it is advised to assign them to a group. This allows for later selection from [context menu of element processing](#) by group name easily.

See also: [Element Editor](#), [Context menu of element processing](#), [Element selection window](#), [Translate window](#), [Stretch window](#)

Mirror window

The window Mirror allows mirroring of one or more [element\(s\)](#) selected from [element list](#). The window is opened from the [context menu of element processing](#) (submenu Rotate or Mirror).

Mirror X	Selects the X plane as mirroring plane. The position of mirroring plane is defined by the input of X coordinate value
Mirror Y	Selects the Y plane as mirroring plane. The position of mirroring plane is defined by the input of Y coordinate value
Mirror Z	Selects the Z plane as mirroring plane. The position of mirroring plane is defined by the input of Z coordinate value
X, Y, Z	Coordinates of mirroring plane
Mirror	All element(s) selected from element list are mirrored by the mirror plane.
 	Undoes (reverts) the latest action(s) in the given editing context or
Undo (Strg-Z) Redo (Strg-Y)	Redoes (reapplies) the action lately undone in the given context .



The mirroring is applied equally to all elements currently selected in the [element list](#). If there are no elements selected all *Mirror button* is set inactive.

Remark: Mirroring by Z plane is available only in 3D projects.

Remark. To rotate elements of belonging together it is advised to assign them to a group. This allows for later selection from [context menu of element processing](#) by group name easily.

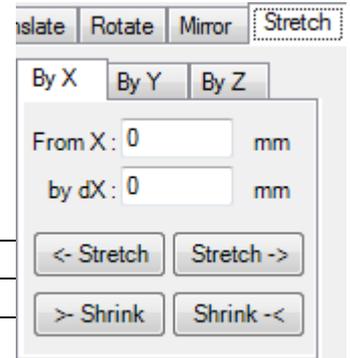
See also: [Element Editor](#), [Context menu of element processing](#), [Element selection window](#), [Translate window](#), [Stretch window](#), [Rotate window](#)

Stretch window

The Stretch window allows stretching (extending, lengthening) of one or many [elements](#) (selected the [element list](#)) located on one side of specified coordinate by specific distance. By entering negative distance the effect of "shrinking" (shortening) elements can be reached.

The window Stretch can be called from the [context menu of element processing](#) (a sub menu Stretch).

By X	Selects stretching operation in X direction.
By Y	Selects stretching operation in Y direction.
By Z	Selects stretching operation in Z direction. Remark: Not available for 2D and 3D-Layered projects .
Stretch->	Translates points of one or many elements (selected the element list) which coordinate values are larger then (are located right to) the From-Entry by specific interval by
<-Stretch	Translates points of one or many elements (selected the element list) which coordinate values are smaller then (are located left to) the From-Entry by specific interval by
Shrink<-	Translates points of one or many elements (selected the element list) which coordinate values are larger then (are located right to) the From-Entry by specific interval -by
->Shrink	Translates points of one or many elements (selected the element list) which coordinate values are smaller then (are located left to) the From-Entry by specific interval -by
  Undo (Strg-Z) Redo (Strg-Y)	Undoes (reverts) the latest action(s) in the given editing context or Redoes (reapplies) the action lately undone in the given context .



The shift is applied equally to all elements selected from the [element list](#), but only restricted to coordinates located on one side of the From-Input value. If no elements are currently selected buttons are inactive.

Remark: Repetitive combinations of "+ Interval" and "- Interval" (in that order) can be used to revert the operation.

Important: Using negative "by" value will in most cases result in irreversible result!

Remark: Z-coordinates of elements in [2D or Layered 3D projects](#) are calculated from layer's thickness. Therefore stretching in Z direction is inactive in such projects.

See also: [Element Editor](#), [Context menu of element processing](#), [Element selection window](#), [Translate window](#), [Rotate window](#), [Mirror window](#)

Slopes and Roundings

Elements are defined in AnTherm by their coordinates – X1, X2, Y1, Y2 and, in 3D, additionally Z1 and Z2. In previous versions this allowed only for rectangular elements. The finite differences method requires rectangles. Still it is now also possible to enter slopes and roundings which are subsequently approximated by rectangular steps. The number of steps can be defined by the user to achieve a good balance between the necessary precision and the computation speed.

To modify the method of input as little as possible we have decided to insert an additional button into the element editor which allows to convert the current element to a slope or roundness. This method considers the coordinates of the two points as the ending points of the slope or roundness. The slope is modelled as a triangle, the roundness as a sector of an ellipse. By combining several rectangular elements and converting them to triangles or sectors of ellipses you can, for instance, represent roofs or camins.

Moreover, it is now possible to input elements rotated by an angle between 0 and 90° (clockwise or counter-clockwise). These elements can be rectangular or triangular and will eventually also be converted to several rectangular elements.

New Elements

The new function „Slopes and Roundings“ permits input of three additional types of elements by first specifying the coordinates of two corner points and then clicking the button “Convert to Slope or Roundness”. This makes AnTherm interpret the new element not as a rectangle but as another type of element. The type of element can be chosen in the window that appears after the click on the button. In the most basic case, there are the following options:

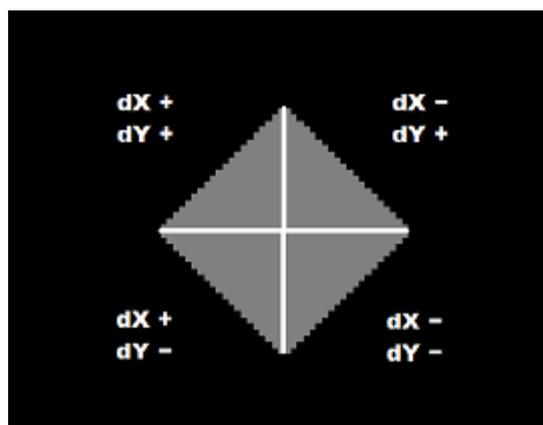
- Slope: Oblique elements have the shape of a triangle.
- Roundness: Roundings have the shape of a sector of an ellipse.
- Roundness Complement: These elements correspond to the difference between a sector of an ellipse and a complete rectangle.
- In the most basic case the chosen shape is drawn with its oblique or round edge ending with the specified corner points.

Moreover, it is possible to draw geometric shapes which are composed of up to four oblique or round elements.

Quadrants

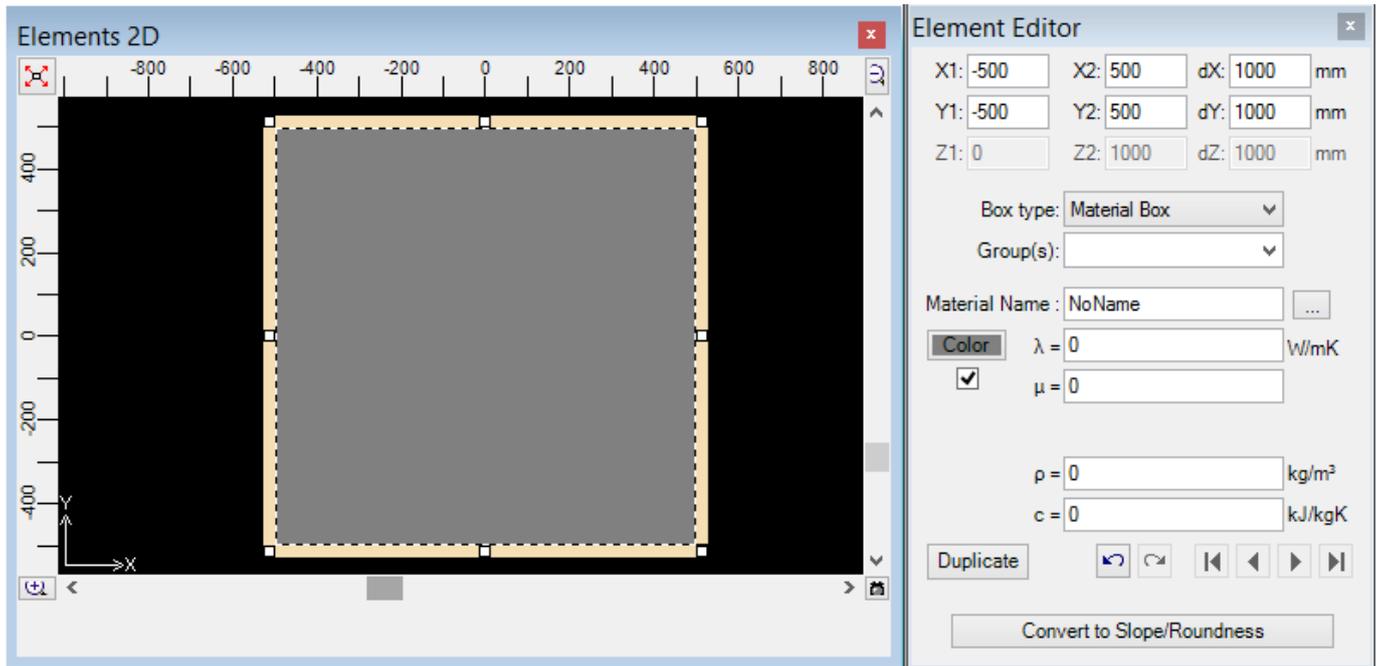
Upon conversion into a slope or roundness the element can be divided into up to four quadrants. This has the effect that up to four elements of the chosen shape are drawn which in turn are approximated by rectangular elements. What is important is that the element is always drawn in such a way that the corner in the respective quadrant remains empty. So if you draw a slope in the top left quadrant, it goes from the bottom left to the top right corner point of the rectangle and the bottom right corner point of the rectangle is the third corner point of the triangle. The top left corner point of the rectangle remains empty.

The following sketch demonstrates this principle by means of an example: A square has been divided into four quadrants and in each of these quadrants a slope has been drawn.

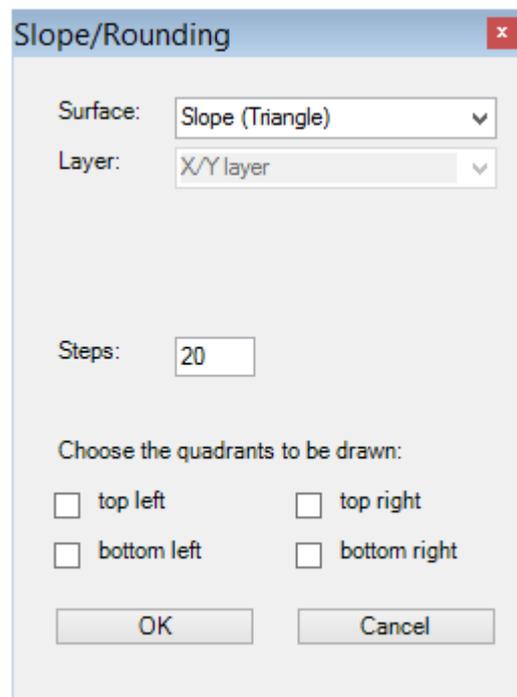


Input

To input a slope or roundness first you have to create a new element:



Next, you click „Convert to Slope/Roundness“. The following window appears:



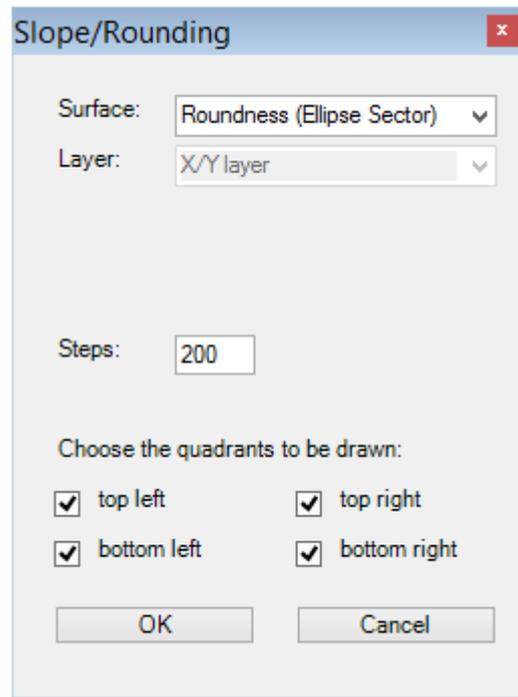
The individual components of this window are rather easy to explain:

- Shape: Slope, Roundness or Roundness Complement
- Layer: Only relevant to the three-dimensional case. Here the user specified whether the slope or roundness should appear in the X/Y layer, the X/Z layer or the Y/Z layer. If the user chooses the X/Y layer, the coordinates Z1 and Z2 remain the same for all of the elements into which the slope or roundness is subdivided. In case of a slope this results in a solid figure with the form of a prism, the base of which is located on the X/Y layer.
- Steps: Number of rectangular elements into which the slope or roundness is effectively subdivided. This number refers to a single quadrant. If you draw a shape composed of the two left quadrants, this shape will be effectively subdivided into the twofold number of steps. Choosing the two upper quadrant also results in a twofold number of

rectangles. However, the number of steps effectively corresponds to the number entered as the staircase-shaped elements are vertically aligned.

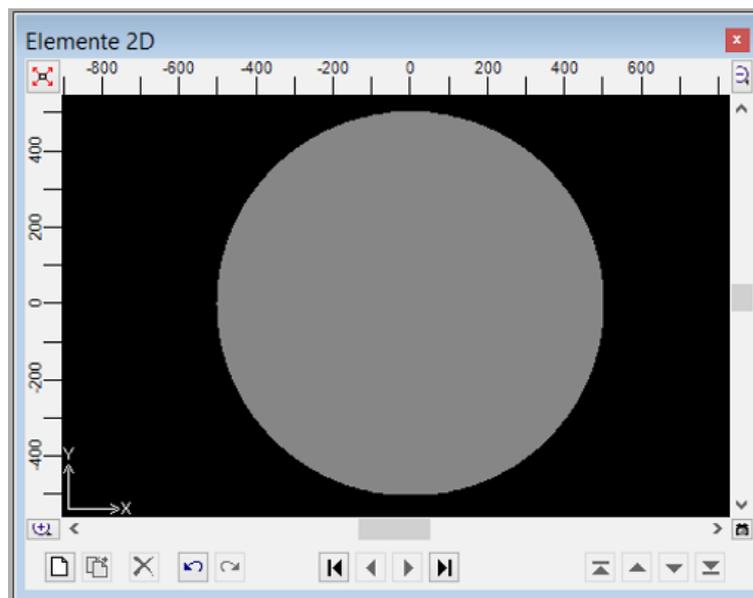
- Quadrants: Here the user can choose the quadrants to be drawn. In the section „Quadrants“ the effect of this has been explained. If no quadrant is specified, the signs of dX , dY and dZ decide how the shape or roundness is drawn.

If we want to convert the square which we initially created to a circle in a high resolution, we have to make the following inputs:

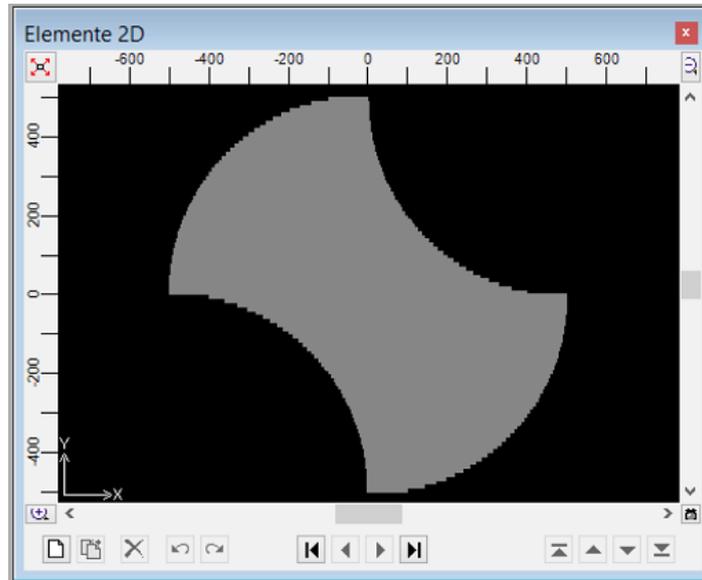


The square is thus subdivided into four quadrants in each of which a sector of an ellipse is drawn. Since the basic element is a square, which means that its width and height are identical, the resulting shape will be a circle. Choosing 200 steps results in a very high resolution (maybe even higher than necessary).

We get to see the result as soon as we click the button „OK“:



It is also easily possible to create composed shapes such as this one:



In order to create this shape we start with a square-shaped material box. We duplicate this box and convert one of the two copies to a roundness where we specify the top left and bottom right quadrants to be drawn. We then convert the other copy to the complement of a roundness and choose the top right and bottom left quadrants. Done!

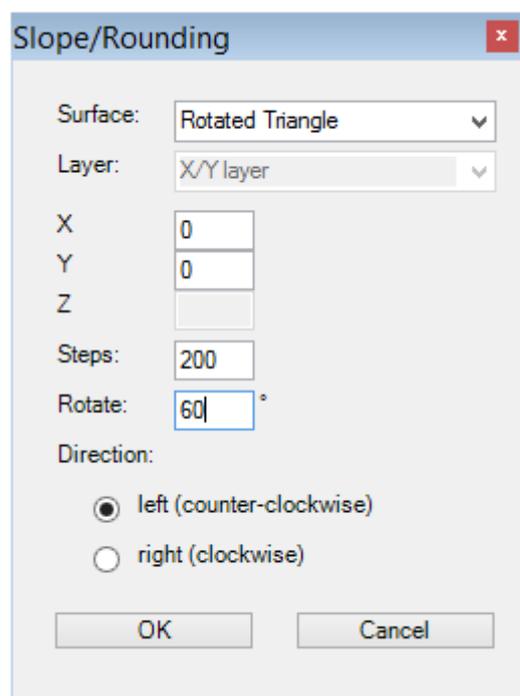
Undo

It is possible to undo the entering of a slope or roundness by pressing CTRL+Z or clicking the respective button in the element window immediately after entering the slope or roundness.

Later on it is possible to remove a slope or roundness by checking the elements belonging to it in the element list (several elements can be chosen at once by pressing the SHIFT key and clicking at the same time or pressing the CTRL key and clicking at the same time) and deleting the elements by pressing the DEL key.

Rotations

Rectangular and triangular elements can also be rotated by an angle between 0 and 90 degrees (clockwise or counter-clockwise). As an example a triangle rotated by 60 degrees is shown here.



Remarks

With increasing number of steps, the precision of the calculation increases. It is recommended to use a high number of steps for optimal precision.

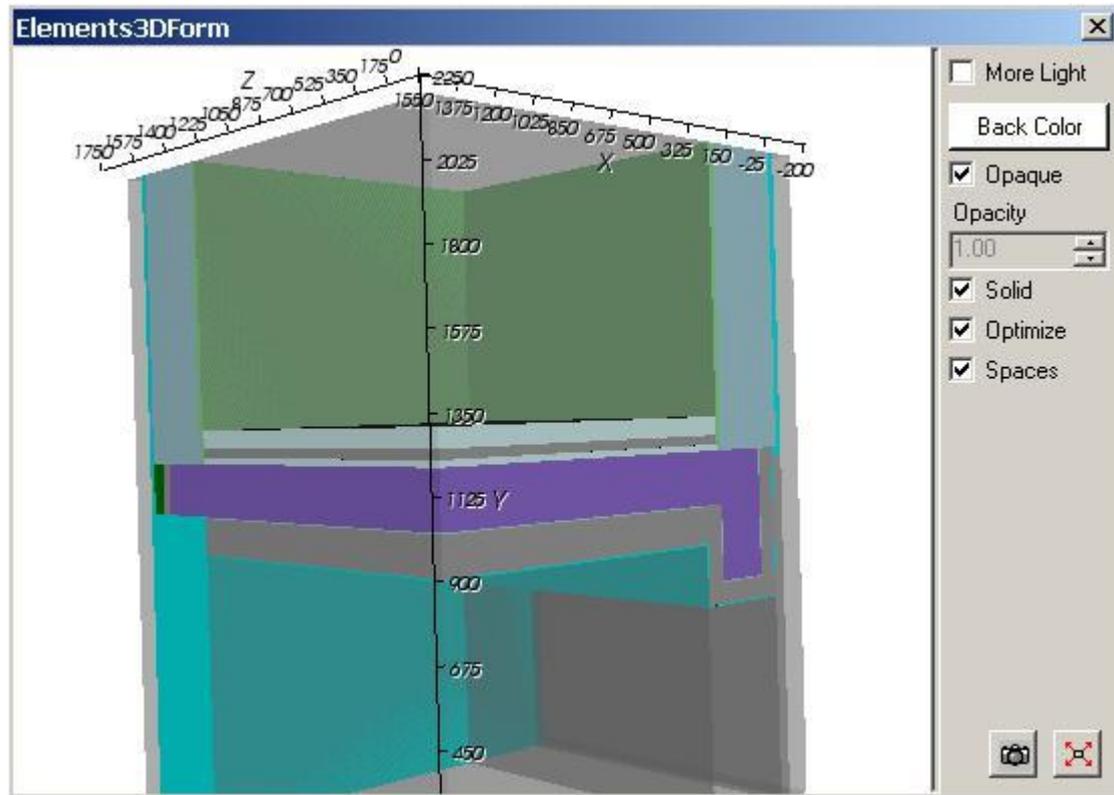
Long, thin elements such as heat sources or thin films require a very high number of steps. We recommend to model these elements as the last ones.

If you want to rotate several elements at once by an arbitrary angle between 0 and 90 degrees, with the rotation point being the same for all elements, it is recommended to follow these steps:

1. Choose all elements you want to rotate. Click the first element, then keep the CTRL key pressed while clicking the next element and so on.
2. Click an element with the right mouse button and select "Convert to Slope/Roundness" in the context menu.
3. Choose "Rotated Rectangle", enter the rotation point and either keep the number of steps suggested by AnTherm or choose a larger number, but not less than suggested.

Elements 3D window

The Elements 3D window shows a 3D view of the component entered. This view is used for continuous verification of the input and of the resulting construction model.



The 3D view can be arbitrarily moved within virtual 3D space with the mouse (rotated, translated or zoomed). The 3D view can be copied onto the clipboard to be pasted later into another application (e.g. Word) from the clipboard - for example for the purpose of integrating this 3D view into some user's report.

More Light	If checked is turns additional "light sources" on to add more contrast or brightness to the view. <u>Remark:</u> Adding more light might result in perception of some shift of how the view coloured - this might be sometimes confusing the the viewer with "overexposure". See also: More Light (setting)
Back Colour	Allow for the choice of the background colour used in the 3D view. See also: Background colour (setting)
Opaque, Opacity	Defines the transparency of model surfaces. AN opaque object is obscure. It obscures fully all objects located behind it. If the opaque checkmark is removed, the opacity can be set to define the degree of transparency of the object. See also: Opaque and Opacity (setting)
Solid	Changing this setting allows for the choice if objects are shown as solid models or as wire frame (lines) only. See also: Solid or wireframe (setting)
Optimize	Coinciding ashlar of the model resulting from overlapping elements (minimum grid) are analysed for equal properties and joined into one object if possible. <u>Remark:</u> Turning this setting on is especially useful to eliminate most of partly transparent planes nested within space areas which might be distracting for the observer.
Spaces	Displaying of space elements (space boxes, shown partly transparent) can be turned on or off.
 Fit	The Position and scaling of the view will be adjusted to show the whole component within the window.
 Copy Image	The image of the 3D view is copied onto the clipboard . The 3D view can be copied onto the <i>clipboard</i> to be <i>pasted</i> later into another application (e.g. Word) from <i>the clipboard</i> - for example for the purpose of integrating this 3D view into some user's report.

The View

The view shows the model resulting from the overlapping of all elements entered within the component (this is the starting point of so called "minimum grid").

How each element is shown depends on [element's type](#) and colour assigned to it, whereas:

- Empty boxes are only visible if selected.
- Space boxes are shown partly transparent.

Element(s) currently selected in the element list are shown emphasized by their edge lines.

Remark: When this window is focused following functions specific to this window are available from the [main menu](#):

- [View→3D Navigation functions](#) (side views, rotations, perspective etc.)
- [Edit→Copy](#) to place the [Image on to the clipboard](#)
- [File→Export→Image...](#) to save the image to picture file
- [File→Export→Scene...](#) to save the [3D scene to 3D scene file](#).

Remark: Labels on axes can be adjusted within [application settings](#).

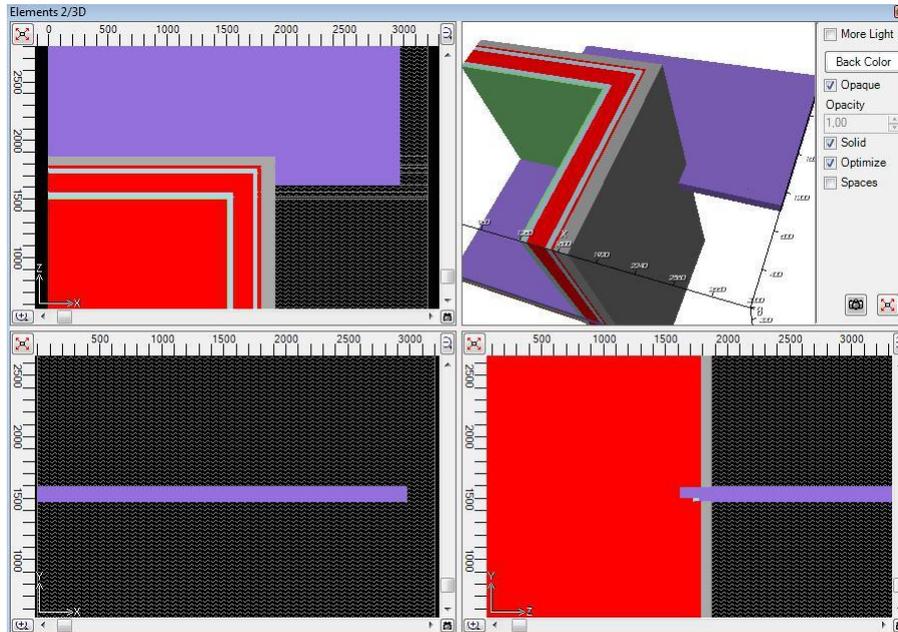
Remark: The [size and proportions](#) of the picture placed onto the clipboard are preset within [application settings](#).

Remark: Current position of the 3D view (the isometry given by the current view vector) will be retained when the window is closed and [saved when the application exits](#). The next time the window is opened the view [isometry will be restored if such behaviour is set within application settings](#).

See also: [Elements 2D window](#), [Element Editor](#), [Element selection window](#), [Further control elements](#), [3D Navigation in 3D windows](#), [TrackBall rotation](#), [The main menu](#), [Coordinate system](#), [Application settings](#)

Elements 23 window

The Elements 23 window is used to input 3D models. It combines three [Elements 2D windows](#) and [Elements 3D window](#) (for example) in one coordinated input form.



Within four quadrants the 3D view is shown at top-right quadrant (by [default setting](#)) and three parallel projections, identical to those of [Elements 2D window](#), are shown in the other three:

- at left-bottom: the X/Y plane
- at left-top: the X/Z plane
- at right-bottom: the Z/Y plane

This provides consistent consistent axe's alignment of neighbouring quadrants connected to the X/Y quadrant (left-bottom).

Each of the three 2D views displays elements in their overlapping order (input order as set within the [element list](#), without any Z-buffer) !

Editing and selection functions are identical to those of [Elements 2D window](#).

The 3D view is identical with [Elements 3D window](#).

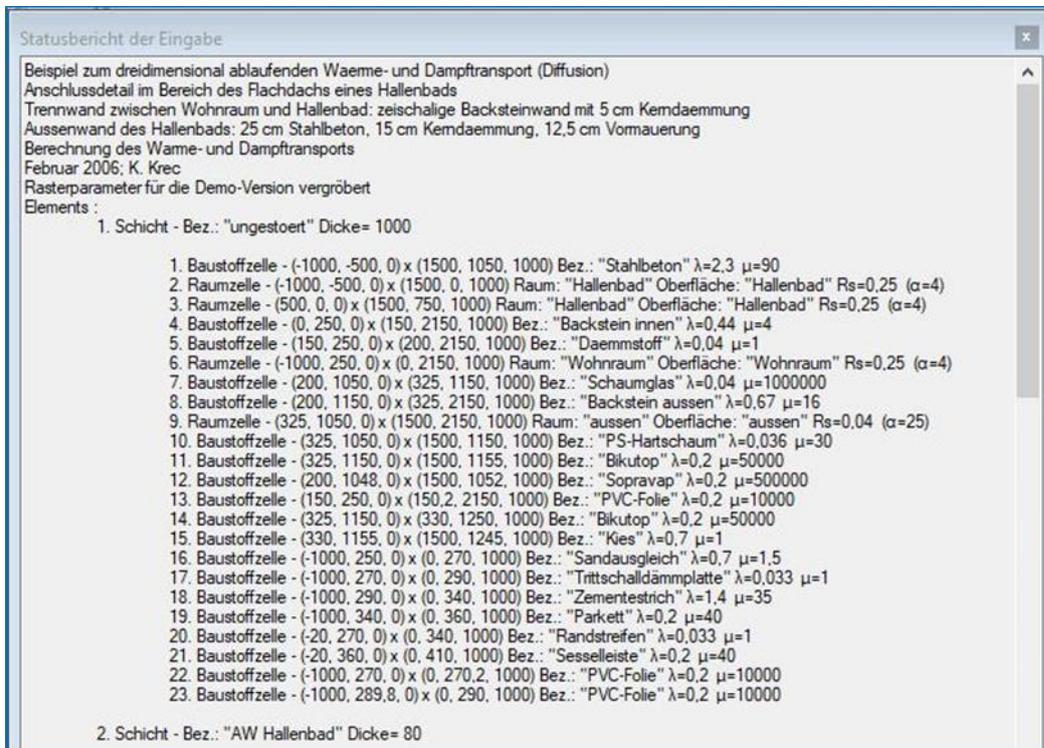
Which view is shown in each quadrant can be defined within [extended user interface settings](#) .

The [views can be copied onto the clipboard](#) to be *pasted* later into another application (e.g. Word) from *the clipboard* - for example for the purpose of integrating this view into some user's report. Views can be [saved to a picture or scene file](#) also

See also: [Elements 2D window](#), [Elements 3D window](#), [Coordinate system](#), [Extended user interface settings](#), [Export Image 2D](#), [Copy Image 2D](#), [Context menu of element editing](#)

Status Report window

The window "Status of input" provides an active and detailed view onto data of the modelled component. This window can be accessed from the [main menu](#) via *View*→*Data Input & Entry*→*Status-Report*.



Contents of this window are permanently updated on each change to input data. Therefore it resembles detailed view on current state of input model.

Under certain circumstances such recurrent updates of this display might lead to significant performance loss of the application (lengthening its response times). Therefore it is advised to open this window for verification purpose and close it when done.

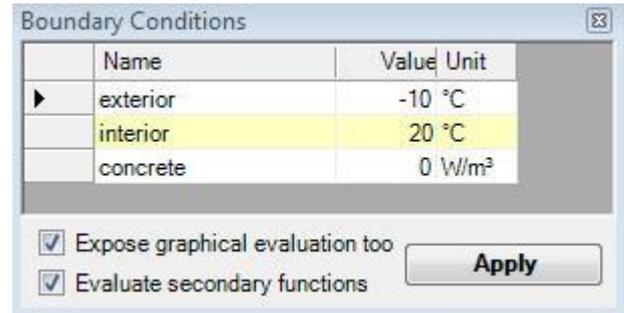
See also: [Reports](#)

Boundary Conditions window

The window *Boundary Conditions* enlists all spaces of which the user sets temperatures and all power sources of which power density can be entered (the boundary conditions). For dynamic, transient problem solution the window mutates to allow [input of time dependant quantities](#).

Temperatures of spaces are initially preset to standard values:

- first space in the list (sorted alphabetically) is set to airspace temperature of -10°C
- all other spaces are set to the temperature of +20°C.



Remark: These standard values can be changed within [application settings](#).

Spaces are enlisted alphabetically. Knowing this, by consequently naming spaces you can target proper automatic assignment of preset values.:

- *Exterior* precedes *Interior*
- *Space 00* precedes *Space 03* and *Space 10*
- *Space 11* precedes *Space 1*.

If the component contains power sources (heat sources or sinks) also, their names will be offered here for the input of their respective power density [W/m³]. Initially power sources are set to have no power (equal 0 W/m³).

Remark: If there is solution of vapour calculation available too, the input of relative humidity of air space, required for vapour evaluations of partial pressure, is also offered in additional dedicated column:

	Name	Value	Unit	Rel.Humidity	Part.Pressure
▶	Room 0	-12	°C	80 %	1.734 hPa
	Room 1	20	°C	53 %	12.386 hPa



Relative humidity of air spaces is initially preset as follows:

- first space is assigned the relative air humidity of 80%
- all further spaces are assigned relative air humidity of 53%.

Remark: These standard values can be changed within [application settings](#).

Name	Name of the space or power source (not editable)
Value	The value of the boundary condition of specific space (temperature °C) or power source (power density W/m ³).
Unit	Unit of the value of that specific boundary condition (not editable)
Rel. Humidity	Relative air humidity in percent
Part. Pressure	Partial pressure of vapour (in hPa) resulting from the temperature and relative humidity values entered (not editable)
Expose graphical evaluation too	If checked here, the graphical evaluation (Results 3D window) will be started after boundary conditions are applied. If this setting is turned off the graphical evaluation (if not open yet) will not be executed automatically. See also Application settings .
Evaluate secondary functions	If checked here, secondary functions (like heat flux, relative surface humidity, etc.) will be calculated for graphical evaluation. If turned off (unchecked) only temperature field will be calculated for Results 3D evaluations. This saves significantly time needed for calculation and reduces memory demand during evaluation. See also Application settings
Apply	Confirms data entered in this window end initiates the superposition followed by various evaluations.

You confirm the data entered by pressing the button *Apply*. Boundary conditions will be applied onto basic solutions (g-values). If calculation of basic solutions is also required it will be automatically initiated also.

Remark: The button "Apply" will blink if there is new solution set available from the [solver](#), but the data entered in this window has not been applied yet to the new solution set.

Remark: The button "Apply" is shown disabled if one or more basic solutions have not been calculated (solved) yet, i.e. they are missing.

The temperature distribution with the construction results from the superposition of respective basic solutions each multiplied by its boundary condition. Results are presented as [text reports](#) ready for printing or to be saved as Adobe PDF, MS.Word or MS-Excel files.

The distribution of temperature and further secondary functions is further used in various graphical [evaluations](#).

Templates of Boundary conditions are saved to [the project file](#) when the project is saved.

Remark: All BCs will be merged by their Names into the projects BC-Template upon apply. This will retain BCs even when spaces or power source are renamed or removed from the project. By that the BCs of other projects can be merged and saved - even such with names which are not yet (or no more) available if the current project.

Important: To retain the values of boundary conditions between program executions [the project must be explicitly saved](#) (project data is not saved automatically when boundary conditions are applied!).

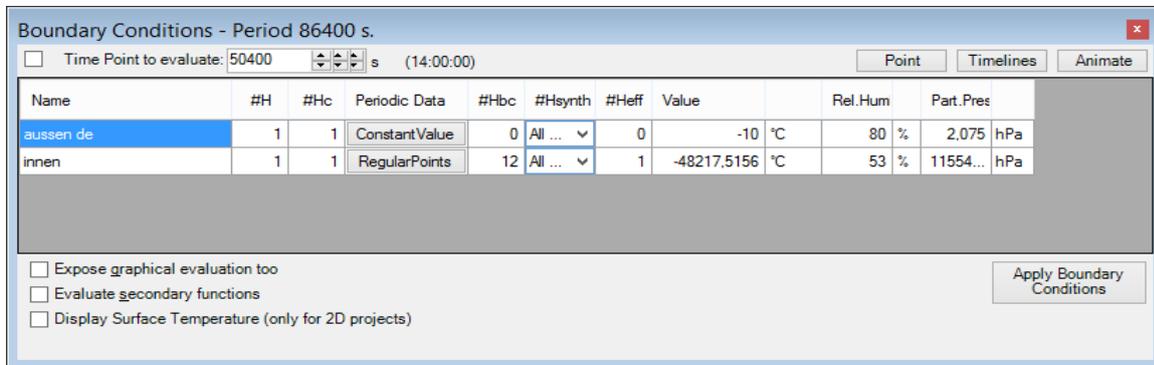
Remark: You will need the value of power density W/m^3 to properly enter the boundary condition specific to power sources. For verification purposes you will find the total volume of each power source within [Modelling Report](#). Within [Results Report](#), in addition to the value of respective power densities, you will also find the overall power of each power source.

VAPOUR-option: Analysis of multidimensional vapour diffusion is only possible with an active VAPOUR-Option of the program..



Transient Boundary Conditions (periodic, harmonic)

For the purpose of dynamic, transient problem (subject to the TRANSIENT-Option) the boundary conditions window mutates to input of time dependant quantities.



TRANSIENT-option: Solving and evaluating time dependant dynamic, periodic problems when heat capacity effects are concerned is only possible with an active TRANSIENT-Option of the program.



The transient boundary conditions are provided as sets of harmonic Fourier coefficients created for respective periodic data (these are managed in [Periodic / Harmonic Data Editor](#)) for each and any Main Period of the transient problem (as selected in [Solver parameter](#) form).

Boundary conditions are kept distinct of each solution period and also saved to [the project file](#) when the project is saved.

Important: To retain the values of boundary conditions lately applied the [project must be explicitly saved](#) (project data is not saved automatically when boundary conditions are applied!).

Remark: All BCs will be merged for each Period (including MainPeriod=0 - steady state) by their Names into the projects BC-Template upon apply. This will retain BCs even when spaces or power source are renamed or removed from the project. By that the BCs of other projects can be merged and saved - even such with periods and names which are not yet (or no more) available if the current project.

Main Period (title-bar)	The Boundary Conditions window title bar displays the Main Period of the currently evaluation transient, periodic problem (as selected in Solver parameter form).
Time Point to evaluate	Some evaluation time point T within the Main Period of the solution. If checked the change to the time point is immediately applied together with boundary conditions and reflected in evaluation windows (Results report , Results 3D window) once these are open.
#H	Number of harmonics solved (harmonic solutions) for the selected Period and available to further Evaluation
#Hc	Number of continuous harmonics solved for the selected Period and available to further evaluation
Periodic Data	Periodic boundary conditions data managed in Periodic / Harmonic Data Editor By default the Constant Value Periodic data will be initially set respectively to the steady state boundary condition.
#Hbc	Number of harmonics available from Fourier Analysis out of periodic boundary conditions data (managed in Periodic / Harmonic Data Editor)_available to further synthesis and evaluation
#Hsynth	Number of harmonics chosen for use in harmonic Fourier Synthesis out of those available from periodic boundary conditions data (#Hbc) The highest number of harmonics used during synthesis (#Hsynth) results from the smallest count of harmonics within the base solution and out of the respective boundary condition and can be further reduced (-1 - all available). Remark: Evaluating in TRANSIENT mode with only 0-th harmonic or only Constant Boundary Conditions is identical to Steady State results (regardless of HARMONIC or TRANSIENT license feature).
#Heff	Effective number of harmonics used in harmonic Fourier Synthesis out of those available from periodic boundary conditions data (#Hbc) and those available from the solution set (#H).
Value	Synthetic boundary condition value at the evaluation time point T
Timelines	Opens Timelines window displaying synthetic temporal, time dependant results at selected Probe Points together with timelines of synthetic boundary conditions applied.
Animate	Displays the Animate Time Dialog used to define temporal animation of transient evaluation by automating changes to the evaluation time point T .

Time Point to Evaluate: provided existence of periodic boundary conditions on can set the evaluation time point (within the main period)

Remark: During graphical evaluation [the time point information](#) will be displayed in the [title of the colour bar](#).

For the evaluation the harmonic base solutions are combined with the respective harmonic coefficients of the boundary condition the belong to (complex linear combination). The resulting distribution harmonic coefficients it then passed for Fourier synthesis for specific [evaluation time point T](#).

Remark: Provided that the periodic boundary conditions did not change, only synthesis for [new time point](#) must be executed.

Remark: Evaluating in TRANSIENT mode with only 0-th harmonic or only Constant Boundary Conditions is identical to Steady State results (regardless of HARMONIC or TRANSIENT license feature).

MULTICORE-option: *Speeding up computationally intensive jobs by distributing them on multiple processors or processor cores for parallel execution is only possible with an active MULTICORE-Option of the program.*



Remark: The input of relative air humidity (when vapour diffusion problem has been solved too) are still constant (steady state) and independent of air temperatures. Only the partial pressure will be recalculated for respective, [evaluation time point](#) dependant, air temperature.

See also: [Element Editor](#), [Element types](#), [Evaluations and Results](#), [Standard Boundary Conditions](#), [Periodic / Harmonic Data Editor](#), [Animate Time Dialog](#), [Timelines Window](#)

Periodic / Harmonic Data Editor

The Periodic / Harmonic Data Editor window provides the means of creating, editing and managing periodic climate data required as boundary conditions to transient evaluation.

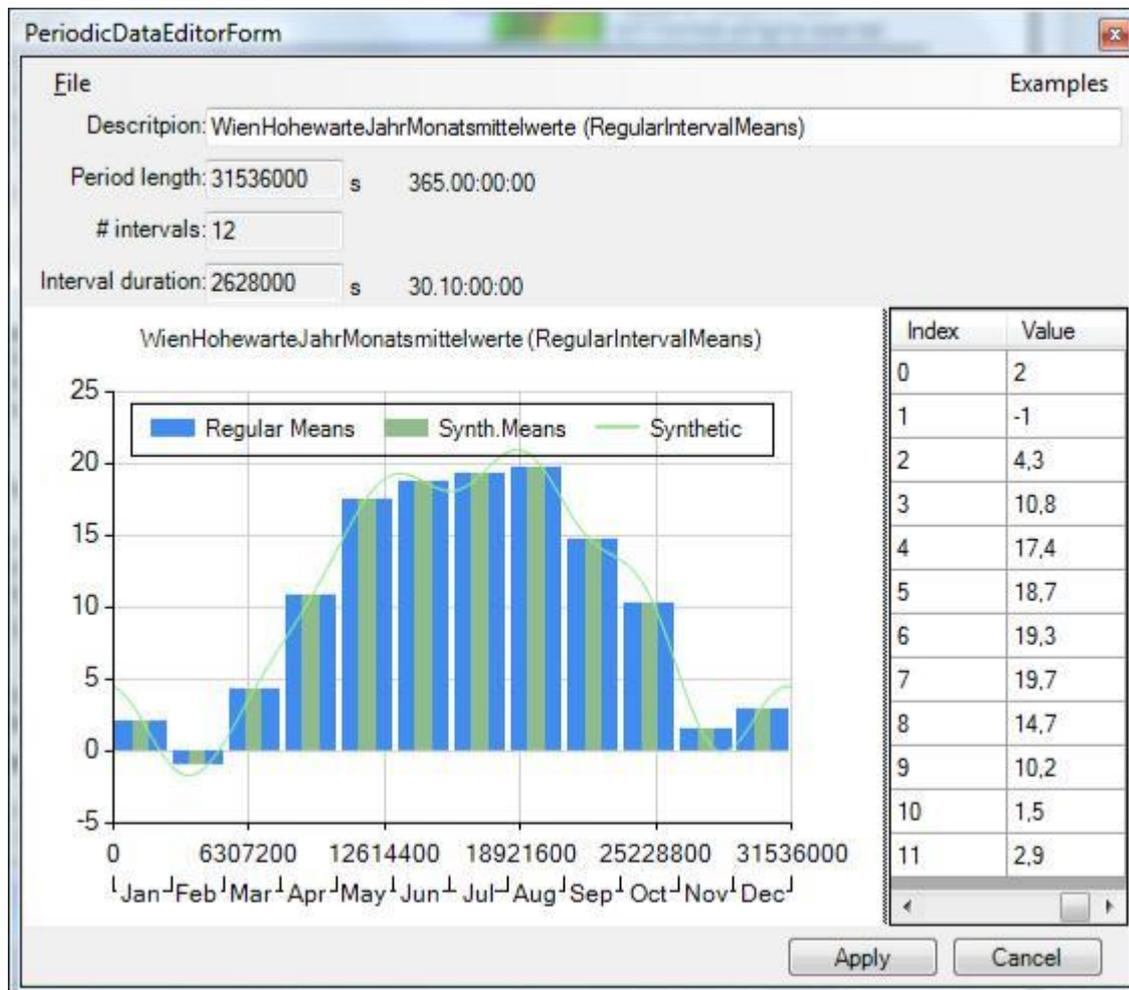
TRANSIENT-option: Solving and evaluating time dependant dynamic, periodic problems when heat capacity effects are concerned is only possible with an active TRANSIENT-Option of the program.



The Periodic / Harmonic Data Editor will be used out of the [Boundary Conditions window](#) when transient problem is to be evaluated and time dependant boundary conditions are needed for that purpose. For convenient managing periodic data sets it is available from a [Tools Menu](#) also.

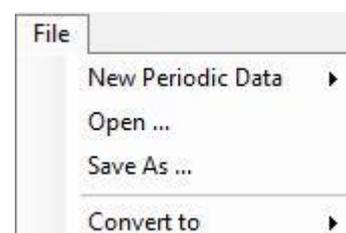
There are following types of data which could be managed within Periodic / Harmonic Data Editor:

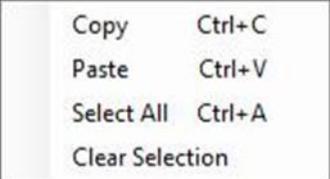
- Constant
- Regular Points
- Regular Means
- Irregular Means
- Irregular Steps
- Harmonic Coefficients.



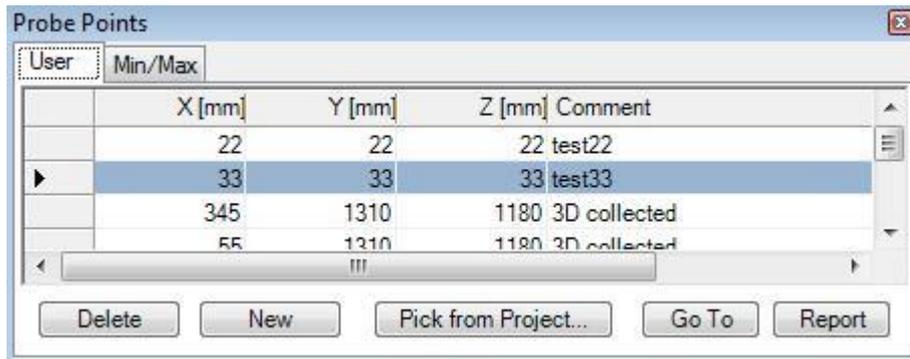
The window contains three parts:

- File & Examples Menu
- Info
- Graph
- Data Grid



File Menu:	<ul style="list-style-type: none"> • New - create new data set of the specific part. A dialog is shown to provide the information required for specific type (see "Info Part"). • Open/Save/Save As... - provides the file access to the data • Convert To... - allows simple synthetic conversions between selected periodic data types.
Info Part:	<p>The Info part displays the characteristic information for the dataset being processed. Depending on the type of data different information is displayed and eventually available to in put.</p> <p>The initial specific values are supplied during the creation of the data set (File->New...) and a limited set of them, depending on the data type processed, can be changed later.</p> <ul style="list-style-type: none"> • Description identifying the data • Period Length • Number of Interval
Graph Chart Part:	<p>The graph chart represents and displays the input data along with its synthetic variant resulting from harmonic Fourier analysis and synthesis.</p> <p>Zoom with lasso...</p> <p>The legend provides the identification of patterns shown. A double click on to the legend will change its position relative to the chart area (right, left, top, bottom).</p> <p>Data points can be selected.</p> <p>Data points (or bars) can be dragged changing the value (press ESC to abort the unintentional drag operation an revert to the previous value).</p>
Context menu of the Graph Chart:	<ul style="list-style-type: none"> • Reset Zoom • Zoom enabled • Zoom X only enabled • Appearance • Page Setup... • Preview... • Print... • Copy • Export Image... • Show synthetic data • Window Function (Gibbs Effect) <p>Selection of the windowing filter function to reduce the Gibbs effect (e.g. while working with switch functions "IrregularIntervalSteps" or dense data with high value changes).</p> <ul style="list-style-type: none"> • Fejer window • Hann window • Hamming window • None <ul style="list-style-type: none"> • Cut-Off Harmonic <p>Limits the number of harmonics for the Fourier syntheis.</p>
Data Grid Part:	<p>The data is input into the Value column.</p> <p>The native Copy & Paste of the selected cell(s) is provided.</p> <p>Capability to paste data from/to the Value column streamlines the input procedure (also available via Context Menu).All numbers contained in the paste buffer (regardless if column(s) or row(s)) will be parsed as double entities (entries in error and not parse able as such in be ignored) and transferred in the order to the value column.</p> 
Buttons:	<p>With the button "Harmonics" the user can open a window where the individual components of the harmonic Fourier series are displayed in their temporal progression. The harmonic value at a given point of time is (approximately) the sum of the individual components. The user can choose which of up to twelve components should be displayed. It is also possible to assign various colours to the components.</p> <p>With the camera button it is possible to create a screenshot which can later be used in the report builder. In the Harmonics window there is this possibility as well.</p>

Probe Points / Extreme (min/max) Locations window

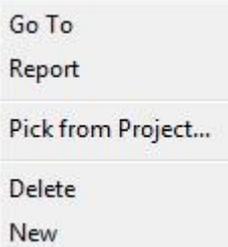


The window *Probe Points* is used to input any number of arbitrary points for which the [Probe Points report](#) shall be generated (second part of [Results report](#)). The list of probe points can be extended from the [Result 3D window \(Probe\)](#) also (by "Collect point").

The Min/Max tab of the window lists interesting locations with extreme values (e.g. points lowest and highest temperatures at surfaces of all spaces and within the interior of all power sources) thus providing the ability to [select them for further evaluation](#). This read only listing is automatically provided and updated as soon as any [boundary conditions have been applied](#) to the solution.

User	Min/Max	X [mm]	Y [mm]	Z [mm]	Comment
		345	1310	1180	Room 1 20°C Tmin=10,43°C
		1372,75	1310	0	Room 1 20°C Tmax=18,11°C
		349	1250	1176	PS 0 0W/m² Tmin=7,1°C
		1525	1250	166 0	PS 0 0W/m² Tmax=-17,64°C

X, Y, Z	Coordinates of the point in millimetres
Comment	An (optional) description of the point entered (used for documentation only)
Delete/New	Deletes the current data line or append a new one
Pick from Project...	Probe points saved within another project will be read and appended to the list.
Report	Confirms the data entry and initiates the creation of probe points in the Results report .
Double click onto data record mark	Initiates translation of the current probe point shown in the Result3D window to the position shown in the line clicked - it translates Slices X/Y/Z accordingly. Within a 2D-project the Z-slice is set to 500 mm.
Go To	Remark: It is data shown in the currently marked line transferred - this means, that a double click onto column or new row will also result in setting the current probe point to values shown selected by data mark arrow.



The input is confirmed by pressing the button *Apply*.

The [second part of Results report](#) will show evaluation of temperatures for all points (x, y, z coordinates) entered in this window. If some point does not belong to the interior of the construction no temperature values will be shown in the corresponding line of the report.

Probe points entered are saved to the [project file](#).

Remark: Similar [evaluation of one probe point](#) is available within [graphical results](#) of the application.

Remark: The active tab will be saved upon the window is closed together with the application settings (*LastSelectedProbePointsFormTabPageName*) and restored when this window is opened again.

Remark: While "Collect Point" (e.g. out of "Results 3D window") is requested the window will switch to the "User" tab where the new probe point will be appended.

See also: [Results report](#), [Boundary Conditions window](#), [Evaluation windows](#), [Probe \(control panel\)](#), [Coordinate system](#), [Slice X, Y, Z \(control panel\)](#), [Timelines window](#)

MULTICORE-option: *Speeding up computationally intensive jobs by distributing them on multiple processors or processor cores for parallel execution is only possible with an active MULTICORE-Option of the program*



Interface AnTherm/SolRad

Interface AnTherm/SolRad

For customers of AnTherm/SolRad there is now the possibility to evaluate the influence of solar radiation and the long-wavelength radiation to and from the external surfaces of a building construction with regard to thermal transmission and temperature distribution.

Since the influences of the radiations are dependent on their orientations, there can be several „exterior spaces“. An exterior space could, for instance, be subdivided into the spaces „south wall vertical“, „west wall vertical“ and „oblique roof“. In the course of the evaluation, three diurnal variations of the radiation air temperature and the related thermal transmission resistances are assigned to these three spaces.

The radiation air temperature is a fictional temperature which is influenced both by the solar radiation and the long-wavelength radiation exchange. If an AnTherm user specifies the radiation air temperature instead of the exterior air temperature, this means that not only the diurnal solar radiation is considered but also the long-wavelength radiation exchange between the exterior surfaces of the exterior building components on the one hand and the sky and the terrestrial environment on the other. The use of the radiation air temperature implies that the absorption and reflection processes occur at the exterior building component surfaces. The used method is capable of exactly reproducing these processes. Until now, the user specified outside air temperatures in AnTherm as exterior boundary conditions. In the stationary case, this is a single value. In the instationary, periodic case this is the 24 values of a diurnal cycle.

SolRad Version 4.1 or higher and **AnTherm Version 9.136.2** or higher are now connected via an interface.

The diurnal variations of the radiations computed by SolRad (global radiations in Wm^{-2}) at the project site can be exported from SolRad for a chosen date and orientation by clicking the „**Export**“ button.

These exported result files can then be imported in AnTherm in order to compute radiation air temperatures and thermal transmission resistances.

Short Instructions Guide:

1. Program SolRad

- 1.1 In SolRad, open the project from which you would like to export the data and click the button „Export“.
- 1.2 Choose date and orientation. The orientation corresponds to an exterior surface of an exterior component appearing in an AnTherm model.
- 1.3 Enter the filename as which the data to be exported should be saved. Mind that you might have to export radiations on differently orientated exterior building components.
- 1.4 Confirm the export by clicking „OK“.

The data are now exported and are then available for being imported to AnTherm.

Repeat the steps 1.2 to 1.4 until the radiations on all exterior building component orientations that appear in the AnTherm model have been exported.

2. Program package AnTherm

The computation of the radiation air temperatures from the outside air temperatures and the solar radiations imported from SolRad can be done for the stationary and for the instationary periodic case. To ensure compatibility with SolRad, it is only possible to calculate with period length being one day in the instationary case.

Important!

The computation of the radiation air temperatures not only leads to a temperature value (in stationary case) or a diurnal cycle of the temperatures (in the periodic case) but also a thermal transmission resistance which has to be assigned to the considered exterior space.

Since the thermal transmission resistances have been entered in AnTherm when the geometric modelling happens but the assignment of the boundary conditions only happens in the course of the evaluation (the actual computation), it is necessary to compute the radiation air temperatures before the geometric modeling, either by using the tool „Radiation Temperatures Calculator“ or the „Periodic/Harmonic Data Editor“ in the menu „Tools“. The thermal transmission resistances assigned to the radiation air temperatures have to be linked by the user to the exterior spaces in the course of the geometry input.

2.1 Stationary Computation

- 2.1.1 Choose the „Radiation Air Temperatures Calculator“ in the menu „Tools“.
- 2.1.2 Into the field „Outside Air Temperature“, enter the diurnal average of the exterior air temperature for the day for which SolRad computed the radiations. Usually this value matches the long-year monthly average at the project site.
- 2.1.3 Into the field „Environment Air Temperature“, enter the diurnal average of the surface temperature of the terrestrial environment. In almost all cases this value matches the diurnal average of the outside air temperature. The exceptions are cases with heavy snow or bodies of water.
- 2.1.4 Into the field „Global Radiation“, enter the diurnal average of the area-related global radiation to the exterior surface of the considered exterior building component in Wm^{-2} . This value can be either manually derived from the program SolRad or automatically imported from the result file by clicking the button „Import“.
- 2.1.4 In the field „Absorption of Sun Radiation“ the color of the exterior surface of the considered exterior building component is classified. The absorption number specifies the percentage of the solar radiation hitting the surface that is absorbed; the remainder is

2.2 Periodic Computation

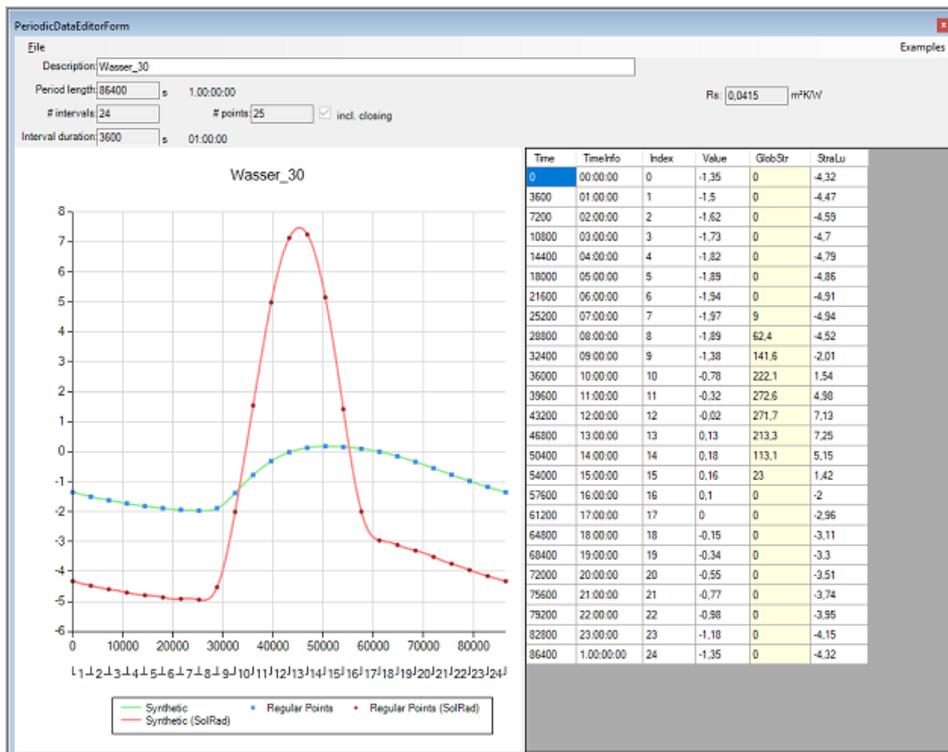
- 2.2.1 Choose the „Periodic/Harmonic Data Editor“ in the menu „Tools“.
- 2.2.2 After clicking „File“ you can either enter the outside air temperature again („New Periodic Data“) or read the already entered and stored diurnal cycle („Open ...“). If you enter the data anew, you have to choose „Regular Points“. The period length has to be a day (86400 sec) and the number of intervals („#intervals“) 24. With these settings the input of the hourly values of the outside air temperature is settled.
- 2.2.3 The import of global radiations calculated by SolRad in advance is done by selecting „File“ and then „Import“ and „SolRad“ once again.
- 2.2.4 The assignment of the color occurs in the next window. Regarding the significance of the input the things said for 2.1.4 apply here in an analogous fashion. By clicking „Import“, the diurnal cycle of the radiation air temperatures is calculated.
- 2.2.5 The thermal transmission resistance assigned to the diurnal variation of the radiation air temperature has to be written down and, in the course of entering the geometry, assigned to the exterior space.
- 2.2.6 By choosing „File – Save as“ the computed diurnal cycle of the radiation air temperature is saved along with the diurnal cycle of the outside air temperature. Thus it can be used for evaluation after the modelling of the geometry and the computation.
- 2.2.7 In the course of the evaluation the column „Periodic Data“ is to be chosen in the window „Boundary Conditions“. Via „File – Open“ the diurnal cycles of outside air temperature and radiation air temperature are imported upon choosing the filename determined in 2.2.6.
- 2.2.8 As a boundary condition for the chosen exterior space, the entered diurnal variation of the air temperature („Apply“) or the computed diurnal variation of the radiation air temperature („Apply StraLu“) can be chosen. If the latter has been chosen, AnTherm will check whether the thermal transmission resistance entered for the exterior space matches the computed value. If that is not the case, a warning message is displayed.

Radiation Air Temperatures Calculator

θ_e	Outside Air Temperature	20 °C
θ_u	Environment Air Temperature	25 °C
J	Global Radiation	55,3 W/m ²
α_{sr}	Absorption of Sun Radiation	Bright (0.3) <input type="checkbox"/> Average (0.6) <input type="checkbox"/> Dark (0.9) <input checked="" type="checkbox"/>
a_u	Environment Radiation Percentage	0,25

T*	Radiation Air Temperature	19,6448 °C
Rs,e	Heat Transfer Resistance	0,0398 m ² K/W

Radiation Air Temperatures Calculator



Periodic/Harmonic Data Editor

Psi-Value Calculator (Tool)

The Psi-Value Calculator is used for rapid calculation of the **linear thermal transmittance** of the thermal heat bridge (correction factors Psi (Ψ , L_{ψ} ,) resulting from the input of U-values and lengths specific to the building component concerned and its [thermal coupling coefficient L2D](#) as calculated within a steady state, [two dimensional thermal heat bridge simulation](#).

Calculating U-values needed is supported by the [U-value Calculator](#) integrated within this application.

U-Value	The U-Value specific the the components part concerned (as calculated with a U-Value Calculator , for example). <u>Remark:</u> If specific U-Value is provided by a layered construct entered within a U-Value calculator , the checkmark left to the button "Layered Construct" is marked checked and this field is marked read-only. To override the value manually uncheck the checkmark left to the button "Layered Construct".
Length	The length to be used for the respective U-Value above
Coupling Coefficient L^{2D}	The thermal coupling coefficient specific to the component under consideration (as calculated and reported within the Coupling Coefficients report , for example).
U * l	The result of the multiplication of respective values of U and l.
Total U * l	The sum of respective U * l multiplications.
Psi-Value (Ψ)	The resulting Psi-Value (Ψ , L_{ψ} ,); the linear thermal transmittance

Up to two U-Value and Length pairs can be entered. The formulas used for this calculation are defined in the EN ISO 10211.

Remark: Obtaining the thermal coupling coefficient L2D can be accomplished with an appropriate two dimensional, steady state, thermal bridge heat transfer simulation software. [AnTherm provides class A precision method for such calculation](#).

Calculating U-Values with U-Value Calculator

To calculate respective U-Value with an integrated [U-Value Calculator](#), press the button "Layered Construct", build or load the layered construct within the [U-Value Calculator](#) which is shown thereafter and confirm the calculated result by pressing the Ok-button therein. The calculated U-Value of a layered construct is shown in the U-Value field which is then marked read-only. The checkmark left to the button "Layered Construct" confirms, that the value shown is based on respective layered construct.

The actual data of a layered construct can be changed at any time by pressing the button "Layered Construct" again and editing the data within the U-Value Calculator shown thereafter.

Remark: The actual data of a layered construct is retained until the form is closed. To save the data of each respective layered construct use the "Save" function within the [U-Value Calculator](#).

Remark: You can drag & drop any layered construct (as it is available from a [U-Value Calculator](#) if it is run separately) onto the one of U-Value / Length blocks in this window also.

Remark: To override the value manually uncheck the checkmark left to the button "Layered Construct".

Remark: If no data of a layered construct has been applied yet, the checkmark left to the button "Layered Construct" is shown greyed.

Entering the Thermal Coupling Coefficient L^{2D} obtained from AnTherm

AnTherm's "Coupling Coefficients Report" provides the matrix of thermal coupling coefficients resulting from steady state simulation of respective modelled construction. As AnTherm displays both, the interior and exterior calculated values for precision validation purposes, you can either use any one of the two or apply the mean value of both for example, provided that precision criteria is satisfied.

Important: U-Values and lengths used for the Psi-Value calculation must correspond to the thermal coupling coefficient of the modelled component in the manner defined within EN ISO 10211 and EN ISO 13789.

As soon as simulation results of a two dimensional calculation are available or created in AnTherm you can pick the value of thermal coupling coefficient from the [coupling coefficients report](#) manually.

Remark: The model is interpreted as two dimensional if it is homogenous in the Z direction (the grid contains only one layer in Z). You shall transfer a median value of the two corresponding matrix elements divided by the thickness in Z (within a 2D project it is 1 meter) and rounded according to the close-up error.

Remark: The [project specific Psi Value Determination Form](#) will pick values automatically as soon as calculation results are available.

Entering U-Values obtained from AnTherm

AnTherm's "Modelling Report" shows U-Values for all layered constructs at adiabatic cut-off planes of the modelled component.

Important: U-Values and lengths used for the Psi-Value calculation must correspond to the thermal coupling coefficient of the modelled component in the manner defined within EN ISO 10211 and EN ISO 13789.

Remark: The [project specific Psi Value Determination Form](#) will pick values automatically as soon as calculation results are available.

Additional remarks

- The standalone version of the tool can be invoked with an argument of "en" or "de" to work in the language different from the operating system default.

See also: [U-Value Calculator \(Tool\)](#), [Coupling Coefficients and Precision report](#), [Project types](#), [Psi-Value Determination](#), [Modelling report](#)

U-Value Calculator (Tool)

The window of U-Value Calculator is used for rapid calculation of U-Values for layered constructs (the R-Value is calculated too).

Data of the layered structure can be saved into a layered construct file (<name>.LayeredConstruct). Data can also be loaded from material layers file (<name>.MaterialLayers) or layered construct file (<name>.LayeredConstruct).

This tool can be started from the [menu Tools](#).

Description	d [mm]	λ [W/mK]	R [m²K/W]
Exterior plaster	5	0,8	0,006249999
Insulation	120	0,041	2,926829268
Adhesive mortar	5	0,271	0,018450184
Reinf. concrete	175	2,2	0,079545454
Sand cushion	20	1	0,02
Isol. underlayment	30	0,04	0,75
PS 0/Concrete topping	10	1,2	0,008333333
Concrete topping	50	1,2	0,041666666

R-Value: 4,05774 m²K/W
U-Value: 0,246443 W/m²K

A layered construct contains a list of material layers surrounded by two surface (exterior and interior) boundaries.

For the two surface boundaries one can enter its surface heat transfer coefficient or transfer resistance. The [Menu Edit→Surface transfer coeffs](#) provides a choice of [normative standard values](#). Both name fields of exterior and interior surfaces also provide same choices of [standard normative values](#):

Exterior	R [m²K/W]	α [W/m²K]
Room 0/Ext. transfer coeff.	0,04	25
EN ISO 6946:1996 Exterior wall (not ventilated)		
EN ISO 6946:1996 Exterior wall, ventilated		
EN ISO 6946:1996 Partition wall		
EN ISO 6946:1996 Wall to unconditioned (cellar)		
EN ISO 6946:1996 Wall to uncond. garret		
EN ISO 6946:1996 Wall connected to ground		

Remark: It is also possible to *drag-drop* values from the project specific [surface list](#).

Material layers are input by the entry of their heat conductance and layer's thickness. Based on this values the program calculates heat resistance for each material layer.

Remark: It is also possible to *drag-drop* values from the project specific [material list](#) or from the [material database](#).

The sum of resistances of all material layers and both surface transfers is displayed in the **R-Value** field.

The reciprocal value of the sum of resistances of all material layers and both surface transfers is displayed in the **U-Value** field.

Actually selected data line is marked by an arrow symbol at its left (data record mark). Multiple selection is also possible (e.g. for deletion of line which can be initiated with DEL key).

The complete layered construct can be placed onto the clipboard by the menu function [Edit→Copy](#) to be [pasted](#) later.

Remark: Changing the construct might be disabled if the U-Value Calculator is shown in read-only mode (e.g. when called from the [Psi-Value Determination](#) window).

Tool specific control elements

Insert	Inserts a new layer at the top of material layers list
Insert Before	Inserts a new layer before the one currently selected
Insert After	Inserts a new layer after the one currently selected
Append	Appends a new layer at the bottom of material layers list
Move Up	Moves the selected data line one position towards top of material layers list
Move Down	Moves the selected data line one position towards bottom of material layers list
Delete	Deletes the currently selected data line from material layers list Tip: To delete data line selected press the DEL key Remark: Pressing DEL key will delete all selected lines
Copy to Clipboard	A copy of current layered construct (all layers and the result) are placed onto the clipboard. Data placed onto the clipboard are available there in different formats: <ul style="list-style-type: none"> ☑ complete layered construct, which can be pasted back into U-value calculator, ☑ tabulated text data of layered construct available for insertion into another application (z.B. MS Excel). See also: Menu Edit→Paste, Edit→Copy

Even if the order of layers and the inside/outside orientation do not matter for the calculation of the U-Value one can reorder layers within this tool as needed.

Tool specific menus

Menus specific to this tool are combined into the [main menu](#) of the application:

File→Layered Construct→New	Creates a new, empty, structure of a layered construct. Warning: Save data prior to requesting this function! Remark: Changing the construct might be disabled if the U-Value Calculator is shown in read-only mode (e.g. when called from the Psi-Value Determination window).
File→Layered Construct→Load...	Allows loading data of material layers (<name>.MaterialLayers) or layered construct (<name>.LayeredConstruct) from a file. Warning: Save data prior to requesting this function! Remark: Loading the construct might be disabled if the U-Value Calculator is shown in read-only mode (e.g. when called from the Psi-Value Determination window).
File→Layered Construct→Save As.	Allows storing the data of a layered construct into separate file (<name>.LayeredConstruct).
Edit→Copy	A copy of current layered construct (all layers and the result) are placed onto the clipboard. Data placed onto the clipboard are available there in different formats: <ul style="list-style-type: none"> • complete layered construct, which can be pasted back into U-value calculator, • tabulated text data of layered construct available for insertion into another application (z.B. MS Excel). • Tip: To copy the resulting U-Value to the clipboard alone use the context menu of the respective result field (you reach this menu by pressing the right mouse button over the field).
Edit→Paste	The construct currently on the clipboard can be pasted (inserted) into the current one.
Edit→Surface transfer coeffs→...	Provides a choice of normative standard values of surface transfer coefficients designed for interior and exterior surfaces. The list offers standard value pairs from EN ISO 6946:1996, EN ISO 10077:2003 and EN ISO 13788:2001.

Surface transfer coeffs. ▶	EN ISO 6946:1996 Exterior wall (not ventilated)
	EN ISO 6946:1996 Exterior wall, ventilated
	EN ISO 6946:1996 Partition wall
	EN ISO 6946:1996 Wall to unconditioned (cellar)
	EN ISO 6946:1996 Wall to uncond. garret
	EN ISO 6946:1996 Wall connected to ground
	EN ISO 6946:1996 Roof (not ventilated) - HS up
	EN ISO 6946:1996 Floor to exterior (not ventilated) - HS down (passage, projecting component)
	EN ISO 6946:1996 Roof, ventilated - HS up
	EN ISO 6946:1996 Floor to exterior, ventilated - HS down (passage, projecting component)
	EN ISO 6946:1996 Intermediate ceiling (betw. cond. spaces)
	EN ISO 6946:1996 Floor to unconditioned (cellar) - HS down
	EN ISO 6946:1996 Ceiling to uncond. garret - HS up
	EN ISO 6946:1996 Slab at ground
	EN ISO 10077:2003 Window vertical $\pm 30^\circ$ (planar surface)- HS hor. $\pm 30^\circ$
	EN ISO 10077:2003 Window horizontal 0° - 60° (planar surface)- HS vert. $\pm 60^\circ$
EN ISO 10077:2003 Window red. radiation/conv. (corners and butt joints) - HS hor.	
EN ISO 13788:2001 Walls and slabs etc. (assessing mould and condensation risk)	
EN ISO 13788:2001 Window panes and frames (assessing mould and condensation risk)	

Standardised surface heat transfer coefficients

Depending on the assessment criteria respective surface resistance values shall be used or replaced if calculation target has changed.

The standards EN ISO 6946, 13788 and 10077 provide set of such standardized values.

Description	R _{si} [m ² K/W]	R _{se} [m ² K/W]
Calculating heat losses due to transmission (by component and heat flow direction):		
EN ISO 6946:1996 Exterior wall (not ventilated)	0.13	0.04
EN ISO 6946:1996 Exterior wall, ventilated	0.13	0.13
EN ISO 6946:1996 Partition wall	0.13	0.13
EN ISO 6946:1996 Wall to unconditioned (cellar)	0.13	0.13
EN ISO 6946:1996 Wall to uncond. garret	0.13	0.13
EN ISO 6946:1996 Wall connected to ground	0.13	0.00
EN ISO 6946:1996 Roof (not ventilated) - HS up	0.10	0.04
EN ISO 6946:1996 Floor to exterior (not ventilated) - HS down (passage; projecting component)	0.13	0.13
EN ISO 6946:1996 Roof, ventilated - HS up	0.10	0.10
EN ISO 6946:1996 Floor to exterior, ventilated - HS down (passage; projecting component)	0.17	0.17
EN ISO 6946:1996 Intermediate ceiling (betw. cond. spaces)	0.13	0.13
EN ISO 6946:1996 Floor to unconditioned (cellar) - HS down	0.17	0.17
EN ISO 6946:1996 Ceiling to uncond. garret - HS up	0.10	0.10
EN ISO 6946:1996 Slab at ground	0.17	0.00

Calculating heat losses due to transmission through transparent components (windows):		
EN ISO 10077:2003 Window vertical $\pm 30^\circ$ (planar surface)- HS hor. $\pm 30^\circ$	0.13	0.04
EN ISO 10077:2003 Window horizontal 0° - 60° (planar surface)- HS vert. $\pm 60^\circ$	0.10	0.04
EN ISO 10077:2003 Window red. radiation/conv. (corners and butt joints) - HS hor.	0.20	0.04
Assessing the Mould and Condensation Risk (by component type):		
EN ISO 13788:2001 Walls and slabs etc. (assessing mould and condensation risk)	0.25	0.04
EN ISO 13788:2001 Window panes and frames (assessing mould and condensation risk)	0.13	0.04

Additional remarks

- The standalone version of the tool can be invoked with an argument of "en" or "de" to work in the language different from the operating system default.

Psi-value (window integration)" tool

Integration of the window area into the exterior wall always plays an important role in mathematical evaluation of thermal bridges in buildings envelope. Therefore almost always distinction has to be made between lintels, reveals and sills.

Of course with "Psi-Value Determination (Automatic)" in menu „Tools", there is already a function available that may be suitable to calculate the psi values for window bindings. However, since the offered automatic calculation of U-values in the area of sectional planes is unsuitable for the window, in case of window bindings and/or the consideration of roller shutter boxes, it is recommended to apply the tool, which was specially developed for this purpose. This will not only draw attention to the importance of thermal bridges in the area of window insertions, but also reduce the risk of modeling errors.

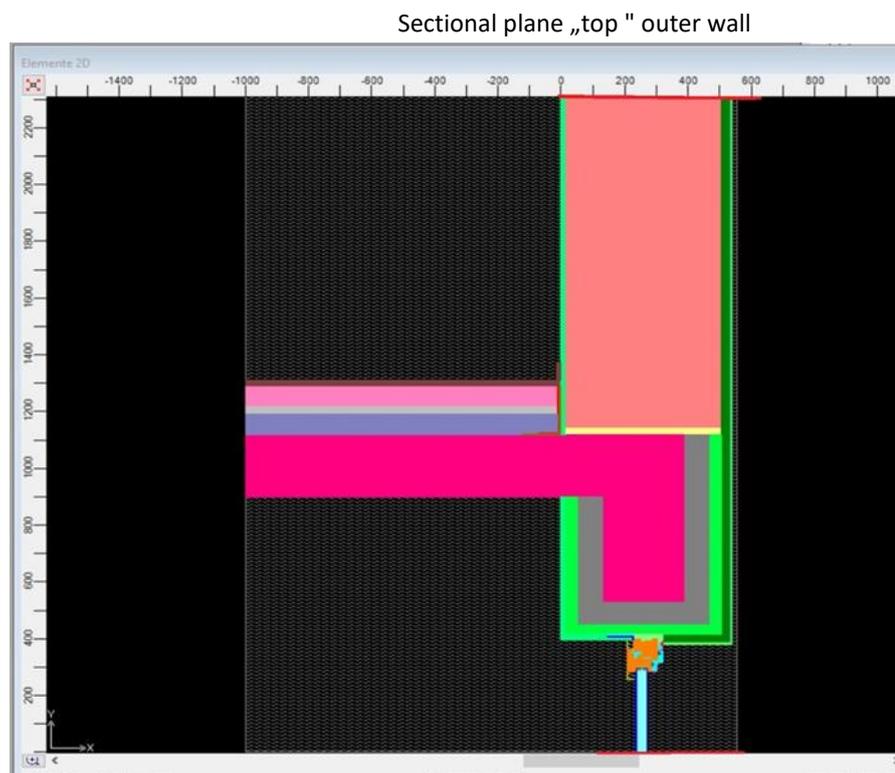
Remark: The automatically calculated U-value for the area of the window will be the U-value of glass and not the window U-value UW for the Psi-value calculation. A confusion between glass U-value and window U-value, e.g. caused by unquestioned application of the automatic U-value calculation, can lead to grossly erroneous results.

The tool "Psi-Value Determination (Window Integration)" differs from "Psi-Value Determination (Automatic)" in that the user is required to enter the window U-value. If this U-value "U FE" is not entered, no report can be created and a corresponding error message is displayed.

For standard calculation, the area of architectural width/ recess has to be used as the window area. Of course, the U-value of the window is used to calculate the coupling coefficient for this area. This approach is used in the calculation of the psi value by setting the "reference point". Also this parameter must be explicitly specified by the user and will be used by AnTherm automatically to calculate the lengths of the two-dimensional model.

Of course both, U-values and associated lengths, must be assigned "correctly". AnTherm suggests an assignment based on entered U-values. This is possible because in the majority of cases, the window U-value will be far greater than the U-value of the exterior wall. If the automatically assignment made by AnTherm is not correct, it is possible to swap the lengths assigned by AnTherm.

The call of the tool which is described here is only permitted if the input area is "cut" in at least on two opposite sides. Of course, it will occur that there will be more/other sectional planes - see the following example of window integration in the lintel area.



Sectional plane "bottom" window

As a special feature, in case of window integration in the lintel area, the model also includes the intermediate ceiling. This means for the interpretation of the Psi-value, that not only the influence of the window connection but also the influence of the ceiling integration into the outer wall is taken into account.

For the case shown above, the window "Psi-Value Determination (Window Integration)" has the following appearance.



The reference point in this example is the edge between the outer surface of the plastered exterior wall (X=540 mm) and the bottom of the (plastered) lintel (Y=380 mm) and is thus compatible with the definition of architectural width/ recess.

U-value of the window was explicitly inserted by the user with 0.95 W/m²K and corresponds to product declaration.

Remark: The automatic determination of the U-value at the lower boundary of the model (section plane "bottom") with the tool "Psi-value determination (automatic)" leads to 0.70 W/m²K and is the U-value of the present triple thermal insulation glass.

Sizes in "Psi-Value Determination (Window Integration)"

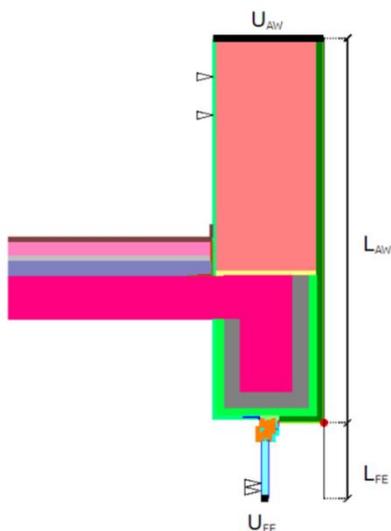
Space as Outdoor space	Select the name that has been assigned to the outdoor space
Thermal Coupling Coefficients 2D	The length-related coupling coefficient of the considered construction is calculated and output data by AnTherm (see also „coupling coefficient report“).The value of the length-related coupling coefficient L2D is predefined as a calculation result and cannot be changed by the user.
U-values	For recalculation to Psi value, the following two U values must always be known: The U value of the window U FE: this U-value must always be entered explicitly by the user and was either determined in a separate calculation or taken from the product declaration. If U-value is not entered, the calculation result makes no sense. In this case, access to the printout of the report remains blocked. The U-value of the exterior wall U AW: This U-value can either be determined automatically from the layered structure along the section line or entered explicitly by the user. In the case of explicit entry, the tick must be removed from "Layered structure" first. Remark: The automatic determination of the U-value of the exterior wall is only useful with a homogeneous layered construction. If there is an inhomogeneous structure - such as a wall made of vertically perforated bricks - the automatically calculated U-

	value changes with the position of the section plane. In this case, the average U-value of the wall must either be determined in advance by a separate calculation or this value must be taken from the product data sheet and entered explicitly.
Lengths	<p>The lengths shown are automatically calculated by AnTherm from the given reference point and cannot be changed by the user.</p> <p>If the automatically assigned lengths by AnTherm are not plausible, it is possible to change them by selecting the arrow to shift.</p>
Reference point	<p>The choice of the reference point, which is to be determined by the user, serves the automatically recalculate of the lengths required for the Psi-value calculation.</p> <p>In general, the reference point must be set in such a way that it correctly represents the fictitious dividing line between the outer wall and the window used in the one-dimensional model in the course of area calculation.</p> <p>If the coordinates of the reference point are not specified, the calculation result becomes nonsensical and the tool for documenting result is blocked.</p>
Psi value (Ψ)	<p>According to the standard, the Psi value is a length-related correction factor intended to compensate the error of the one-dimensional calculation model. This value is the calculation result and cannot be changed by the user.</p> <p>Multiplication of the Psi value with a length leads directly to conductance surcharge in W/K. It is important to know that this length is always measured vertical to the calculation plane of the two-dimensional calculation.</p> <p><u>Remark:</u> In case of the example, this is the width of the lintel. For this particular example, it should be noted that the calculated Ψ-value not only influences the window connection to the wall, but also of the ceiling integration into the exterior wall.</p>



If entries required by the user (U-values, reference point) have been completed, the production of documentation can be ordered by pressing the "Create report" button.

The report has the following appearance as shown here on an example:



$L_{2D} = 0.8246 \text{ W/mK}$	
$U_{FE} = 0.95 \text{ W/m}^2\text{K}$	$L_{FE} = 0.38 \text{ m}$
$U_{AW} = 0.1299 \text{ W/m}^2\text{K}$	$L_{AW} = 1.93 \text{ m}$
$\Psi = L_{2D} - U_{FE} L_{FE} - U_{AW} L_{AW} = 0.2129 \text{ W/mK}$	

Psi-Value Calculation for Three Rooms

AnTherm Version 9.136 has Psi-Value Calculation for Three Rooms (unheated buffer room) as a new tool. It starts with a 2D calculation of a model that thermatically connects these rooms with each other. The temperatures can be either entered separately or a temperature factor can be specified. It is important to enter the rooms in the correct order.

Psi-Value Determination for the Three-Space Case ✖

Space to assume exterior: außen ▼

Another space to assume interior: oben ▼

Another space to assume buffer: unten ▼

U-Value Profiles at adiabatic cut-off planes

ausen <-> oben @ TopBack: (5) ▼ unten <-> oben @ BackLeft: (-2) ▼

Layered construct Layered construct

U 1: 0,10040 W/m²K U 2: 0,10247 W/m²K

Length 1: 2150 mm Length 2: 2521 mm

U * l : 0,21586000 W/mK U * l : 0,25832687 W/mK

Determine Lengths by Reference Point

Leitwerte 2D

L2D 1: 0,218880912 W/mK L2D 2: 0,421230775 W/mK

Temperature Factor

θ ex.: -13,5 K f: 1

θ in.: 20 K

θ bu.: -13,5 K

Ψ value : 0,165924818 W/mK

Determine Reference Point

X: 521 mm at exterior dimensions (außen)

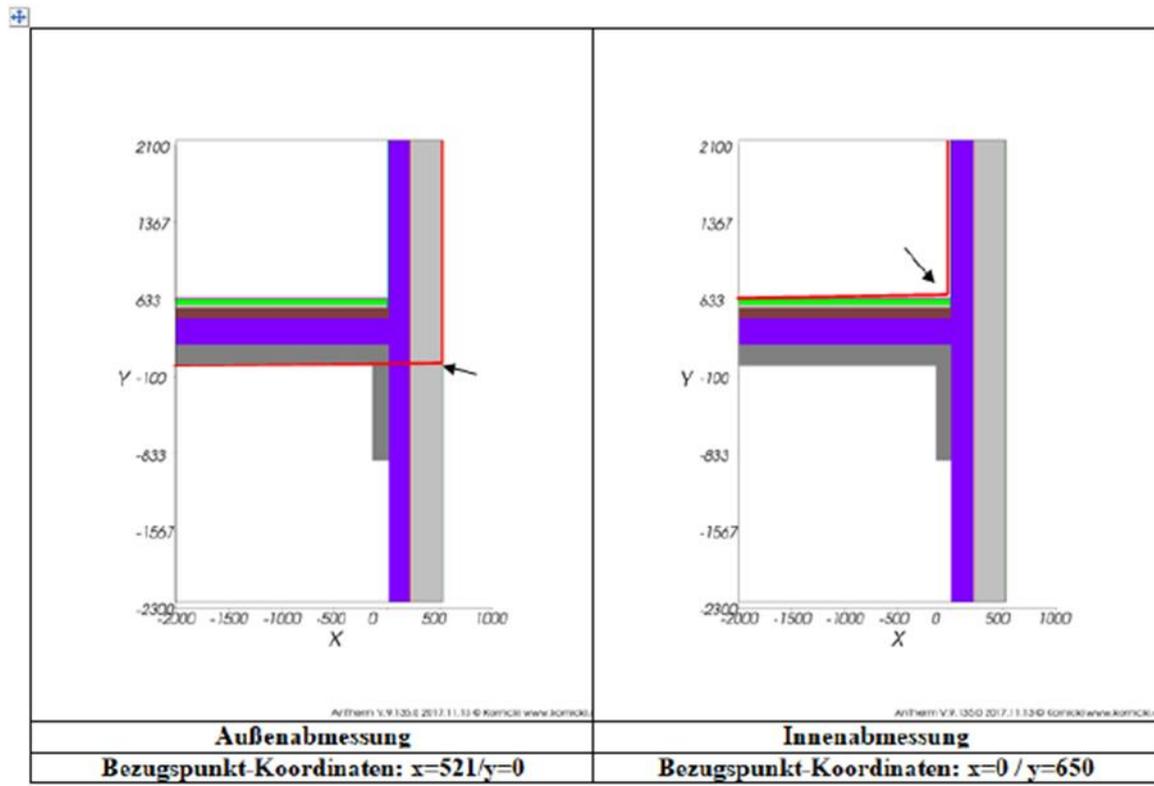
Y: 0 mm at interior dimensions (oben)

by manual input

Generate Report

θ ex. = exterior sp., θ in. = interior (heated), θ bu. = buffer (basement, unheated)

Model - Reference points



Sketch of the calculation process

Assumptions

Space „above“ (top left)	$\Theta_i = 20.0 \text{ } ^\circ\text{C}$	(heated)
Space „below“ (bottom left)	$\Theta_p = 5.0 \text{ } ^\circ\text{C}$	(basement unheated)
Space „outside“ (right)	$\Theta_e = -10.0 \text{ } ^\circ\text{C}$	(outside)

Conductance Values

Conductance „upward-outward“	$L_{i,e} = 0.218881 \text{ Wm}^{-1}\text{K}^{-1}$
Conductance „up down“	$L_{i,p} = 0.421231 \text{ Wm}^{-1}\text{K}^{-1}$

Heat transfer coefficients

EW above	$U_{i,e} = 0.1004 \text{ Wm}^{-2}\text{K}^{-1}$	(from AnTherm layers report)
Cellar ceiling	$U_{i,p} = 0.1025 \text{ Wm}^{-2}\text{K}^{-1}$	(from AnTherm layers report)

Roller Shutter Box Usb Calculator

The tool Shutter Usb Calculator allows for simple and user-friendly modeling of a shutter box geometry in a 2D-model. The goal is to calculate the Usb-value by means of a user interface prompting the user to pass all relevant input parameters. The calculation is in accordance with the EU standard EN ISO 10077.

The input/output of all relevant parameters and results is divided into 3 steps: Geometry, Physics, Calculation.

The screenshot shows the 'Roller shutter calculator' dialog box with the following parameters:

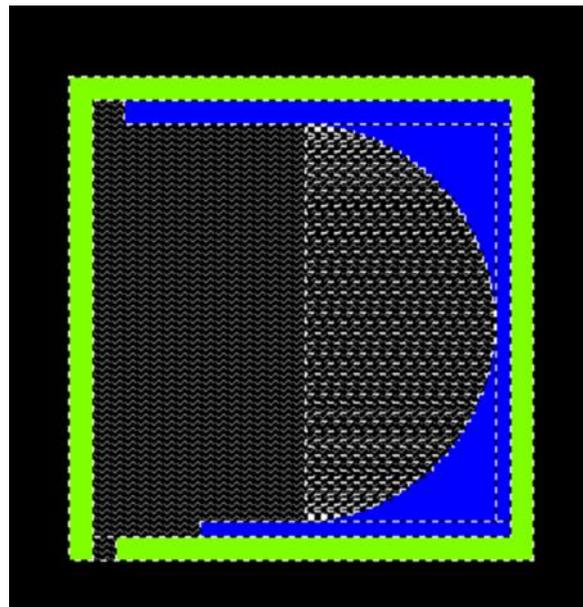
Parameter	Value	Unit
e:	1	mm
e:	2	mm
e:	1	mm
Height:	300	mm
Width:	300	mm
Upper wall thickness:	15	mm
Outer wall thickness:	15	mm
Inner wall thickness:	15	mm
Lower wall thickness:	15	mm
Offset:	30	mm
Thermal insulation	<input checked="" type="radio"/> Definition by thickness	
Upper width:	250	mm
Lower width:	250	mm
Upper thickness:	15	mm
Inner thickness:	10	mm
Lower thickness:	10	mm

In order to create a shutter box, one has to click Tools -> Shutter Usb Calculator. Clicking the button Apply, all set parameters in that tab are accepted and saved. Click Close if you wish to exit the input prompt of the Shutter Usb Calculator.

Afterwards, the shutter can be edited by clicking on the outermost cell of the shutter box (all contours and cells of the shutter box are represented by dashed lines) and click Edit in the Elements window.

The screenshot shows the 'Element Editor' dialog box with the following parameters:

Parameter	Value	Unit
X1:	0	
X2:	300	
dX:	300	mm
Y1:	0	
Y2:	300	
dY:	300	mm
Z1:	0	
Z2:	1000	
dZ:	1000	mm
Box type:	Material Box	
Group(s):	Rollladenkasten1	



The following sections provide a detailed explanation of each input parameter:

Geometry:

First, we introduce the concept of the Basic box. The Basic box is a simple, rectangular standard geometry, which is generated by means of the tool Shutter Usb Calculator. The Basic box features an opening for the shutter at its exterior, bottom edge. Starting from the Basic box, a real shutter box can be created by manually adding cells using the conventional tools of AnTherm. Hence, features can be added, such as notches, profiles, end profiles, additional cells, etc... to model a real shutter box. The benefits of this approach as compared to an entirely manual modeling of a shutter box are:

- Quick and simple approach
- Input parameters for the opening of the shutter in accordance with the terminology of the EU standard
- Automated classification of the ventilation state inside the shutter box
- No explicit modeling of the boundary conditions or adiabatic boundaries needed
- Quick enabling of boundary conditions according to the EU standard
- Quick calculation of the heat flux and the Usb-value

However, the Shutter Usb Calculator is subject to the below restrictions:

- Applicable for simple, rectangular geometries with one cavity, , and in vertical orientation (as displayed in below image).
- Only applicable for 2D-models
- Only one shutter box allowed per file
- A file must not contain cells that do not belong to the shutter box, otherwise the resulting Usb-value might be wrong
- The shutter opening is always part of the construction which we refer to as the Basic box
- The shutter opening is only defined on the exterior bottom edge/face of the shutter box
- The shutter opening settings (defined by the parameters e_1 , e_2 , e_3) can only be edited in the Shutter Usb Calculator tool exclusively (not in the Element window), in order to assure correct treatment of the ventilation condition in the cavity
- The shutter opening is always located on the exterior, which is always on the left of the shutter box
- The default thermal insulation provided by the tool Shutter Usb Calculator is designed for covering the top, bottom and the interior edge of the shutter box cavity with a layer of thermal insulation, having a circular recess in the center..., if required otherwise, needs to be edited manually
- Boundary conditions, such as temperatures and adiabatic boundaries, must not be included manually, they are included in the calculation of the Usb-value by default (the placement of the bottom adiabatic boundary is controlled by the control parameter Offset)
- The model must not contain any heat sources or drains
- The model must not contain any internal boundaries
- Except for the cavity, the model can only contain two more room cells, that is an inner room and an outer room, hence only two temperatures are allowed as boundary conditions (both rooms are created automatically, and must not be created manually by the user)
- The model can only contain two adiabatic boundaries, one being at the top edge of the shutter box all along its entire width, the other being at the lower edge along a width of 60mm starting from the location defined by Offset (both adiabatic boundary conditions are created automatically, and must not be created manually by the user)

The below schematic serves as a guide for an easy understanding of the input parameters listed below.

- Input parameters geometry: parameters that allow for defining a standard geometry (Basic box) for a rectangular shutter box, featuring an opening for the shutter on the exterior bottom edge/face

- e_1 : partial width of the shutter opening (outermost)

Remark: Only edit in the Shutter Usb Calculator

- e_2 : partial width of the shutter opening (shutter itself)

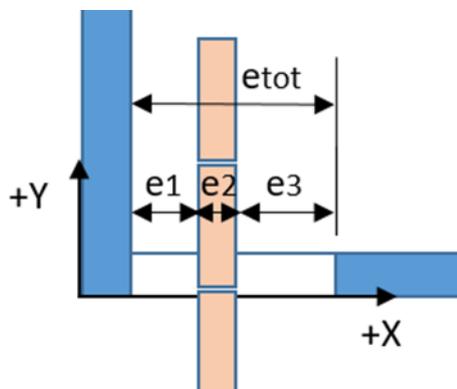
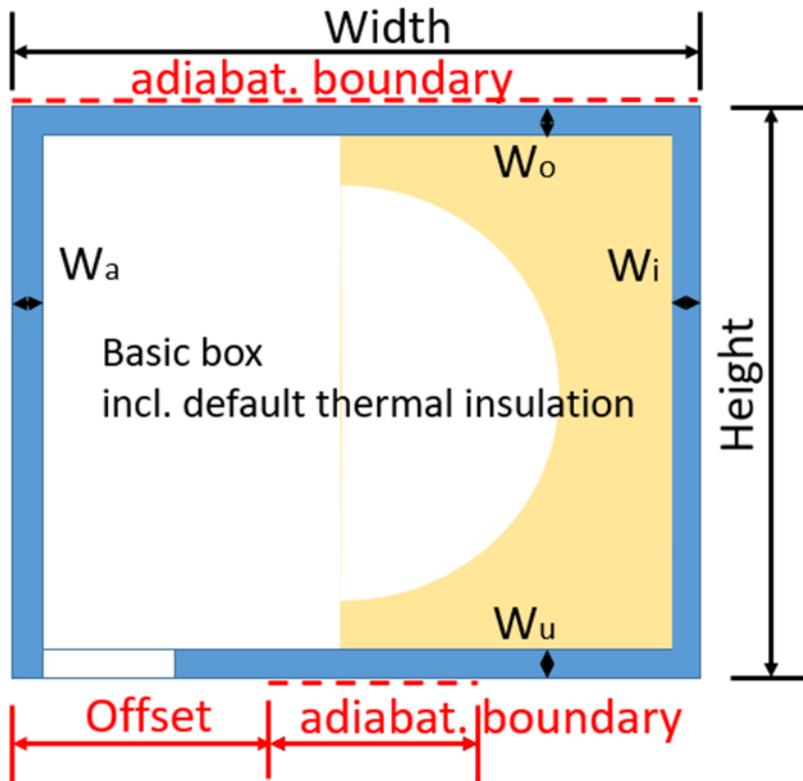
Remark: Only edit in the Shutter Usb Calculator

- e_3 : partial width of the shutter opening (innermost)

Remark: only edit in the Shutter Usb Calculator

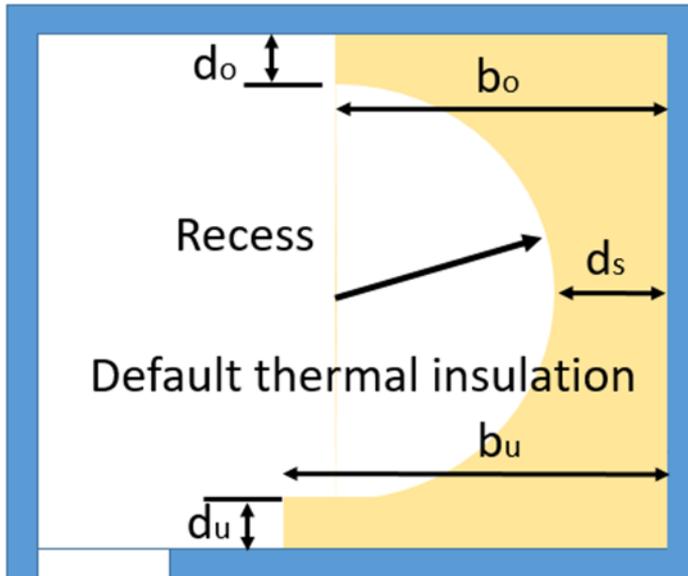
Remark: if there is no information on the shutter thickness itself (width e_2), but only the total width of the shutter opening $e_1+e_2+e_3$, then the user is free to arbitrarily distribute the total width to each partial width..., however please mind the ventilation condition inside the cavity

- Height: height of the shutter box in x-direction
- Width: width of the shutter box in x-direction
- Wall thickness at top W_o , outer W_a , inner W_i , bottom W_u : Wall thickness at each face of the shutter box
- Offset: Desired distance between the exterior edge of the Basic box and the bottom adiabatic boundary (see image below).



The parameters that define the default geometry of the thermal insulation inside of the shutter box are supposed to cover the faces of the cavity (only partially, if desired). Around the center of the cavity we have a recess defined by a circular-shaped cell. In case it is desired to model a customized insulation geometry, the user can manually edit the geometry (add cells) using conventional tools provided by AnTherm. Many industrial products are designed such that the housing itself is manufactured from thermally insulating materials, whereas no internal insulation is included on the inside of the shutter box. In such a case, this component is to be considered as the housing, and in the field Thermal insulation, one needs to select No insulation. In general, that part of the shutter box that defines/contains the shutter opening, is considered as the Basic box.

- Thermal insulation (default insulation): provides the option to either create a default insulation implemented in AnTherm as described above, or alternatively to leave the shutter box without any default insulation
- Width at the top W_o : width of the top layer of insulation in x-direction, whereas the edge towards the exterior side is variable
- Width at the bottom W_u : width of the bottom layer of insulation in x-direction, whereas the edge towards the exterior side is variable
- Thickness at top W_o , bottom W_u , interior side W_i : thickness of the insulation layer at the respective edge inside the shutter box cavity



Physics:

The tab Physics allows for setting/controlling all parameters that are relevant for the calculation of the U_{sb} -value.

The field Box provides the option to select the material for the housing of the shutter box. In fact, this defines the outermost cell of the Basic box, which can be assigned any material from the database.

The field Thermal insulation can only be edited if the user decides to include an (Default) insulation in the Geometry tab in the first place. In that case, the tab Physics provides the option to choose an insulation material for the Default insulation.

Manually added cells can only be edited in their respective Element window.

By clicking OK in the field Environment EU-standard conditions can be enabled. This action assigns boundary conditions (temperatures and thermal transfer resistance of the interior and exterior) according to the EU-standard and the ventilation condition inside the cavity. This extends to the adiabatic boundaries, being in accordance with the EU-standard.

- Material name: defines the material for the housing of the shutter box
- e1+e3 (output only): partial width of the shutter opening, that only includes the two airgaps and does not include the shutter thickness
- e1+e2+e3 (output only): total width of the shutter opening, including both airgaps and the shutter thickness
- The shutter box is (output only): displays the ventilation state of the shutter box cavity, depending on the dimensions of the shutter opening
- Apply EU-standard boundary conditions?: Enables boundary conditions in accordance with the EU-standard
- Interior $R_s(W)$: interior heat transfer resistance
- Temperature: interior temperature
- Exterior $R_s(W)$: exterior heat transfer resistance
- Temperature: exterior temperature
- Cavity $R_s(W)$: heat transfer resistance inside the shutter box
- Temperature: temperature inside the shutter box (please set equal to exterior temperature, if non-EU-standard conditions desired)
- Equivalent Lambda: defines an equivalent Lambda-value for the cavity in case of an unventilated shutter box (to be set by the user).

Calculation:

In the last step, the calculation of the U_{sb} -value can be triggered. Additionally, a list of thermal conductance values is displayed that contribute to the heat flux, which is relevant for the U_{sb} -value.

- Start: triggers the calculation of the heat flux and the U_{sb} -value
- List of thermal conductance values in contact with the interior (output only): a list of all components of the thermal conductance matrix listed in the interior-column
- Report: creates a report that lists all relevant input parameters used for the calculation of the heat flux and the U_{sb} -value
- Total heat flux (output only): displays the total heat flux from the interior through the shutter box
- U_{sb} -value (output only): displays the U_{sb} -value of the shutter box geometry at hand.

Chi-Wert Calculator

Chi-Wert Rechner

Schichtaufbau
 Schichtaufbau
 Schichtaufbau
 Leitwert 3D: W/K

U 1: W/m²K
 U 2: W/m²K
 U 3: W/m²K

* A 1: m²
 * A 2: m²
 * A 3: m²

U * A: W/K
 +
 U * A: W/K
 +
 U * A: W/K
 = Gesamt U * A: W/K

Ψ Rechner
 Ψ Rechner
 Ψ Rechner

Ψ 1: W/mK
 Ψ 2: W/mK
 Ψ 3: W/mK

Länge 1: m
 Länge 2: m
 Länge 3: m

Ψ * l: W/K
 +
 Ψ * l: W/K
 +
 Ψ * l: W/K
 = Gesamt Ψ * l: W/K

* A = Fläche

X Wert: W/K

Projekt speichern

The Chi value can be calculated manually or using the tools already contained in previous versions of AnTherm.

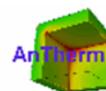
We have included a Chi Calculator in AnTherm.

The Chi value is the point heat transfer coefficient.

You can enter all data manually or make use of the existing tools in AnTherm.

- Leitwert 3D: is automatically computed by AnTherm if the model is a 3D model. Either enter the value manually or do the computations first before opening the Chi Calculator, then AnTherm will take the value from the Thermal Coupling Coefficient report.

Kornicki, Dienstleistungen in EDV und IT
 A-1230 Wien, Othellogasse1/RH 8/2
 Tel/Fax: +43-1-6157099
 email: tkornicki@chello.at



AnTherm
 V.9.136.0 2018.02.06
 (c) Kornicki, all rights reserved

08.10.2018

Thermische Leitwerte [W / K]

Raum\Raum	außen	innen
außen		2,427679
innen	2,427679	

- U value: The U value calculator of AnTherm can be used by clicking the button „Layered construct“.

U-Wert Rechner

Datei Bearbeiten Hilfe

Name:

Aussen R [m²K/W] α [W/m²K]

Aussenluft 0,04 25

	Bezeichnung	d [mm]	λ [W/mK]	R [m²K/W]
	NoName	80	0,04	2
	NoName	300	0,58	0,517241379
▶	NoName	15	0,7	0,021428571
*				

Innen R [m²K/W] α [W/m²K]

Innenluft 0,13 7,692308

R-Wert: m²KW

U-Wert: W/m²K

Kopieren

Einfügen
Einf. Vor
Einf. Nach
Anfügen
Nach Oben
Nach Unten
Löschen

- Area: This must be manually specified by the user.
- Now you get the product $U \cdot A$.
- For the Psi calculation you can use the manual Psi calculator which is included in AnTherm.

Psi-Wert Rechner

Schichtaufbau Schichtaufbau Leitwert 2D: W/mK

U 1: W/m²K U 2: W/m²K

Länge 1: mm Länge 2: mm -

U * l: W/mK + U * l: W/mK = Ges. U * l: W/mK

Ψ - Wert: W/mK

(If you also have 2D-cuts of the model that show the corners, it is possible to use the automatic Psi calculator to obtain the U values and lengths.)

To run the Psi calculator, click the related button.

For the Psi calculation in the manual Psi calculator you also need the **Leitwert 2D**.

For each Psi value a separate calculation has to be performed. By clicking "OK" you copy the result of the computation into the Chi calculator.

Chi-Wert Rechner

Schichtaufbau Schichtaufbau Schichtaufbau Leitwert 3D : 2,307766 W/K

U 1: 0,36918 W/m²K U 2: 0 W/m²K U 3: 0 W/m²K

* A 1: 3,762 m² * A 2: 0 m² * A 3: 0 m²

U * A : 1,38885516 W/K + U * A : 0 W/K + U * A : 0 W/K = Gesamt U * A : 1,38885516 W/K

Ψ Rechner Ψ Rechner Ψ Rechner

Ψ 1: 0 W/mK Ψ 2: 0 W/mK Ψ 3: 0 W/mK

Länge 1: 0 m Länge 2: 0 m Länge 3: 0 m

Ψ * l : 0 W/K + Ψ * l : 0 W/K + Ψ * l : 0 W/K = Gesamt Ψ * l : 0 W/K

* A = Fläche X Wert : 0,918911 W/K

Projekt speichern 

Psi-Wert Rechner

Schichtaufbau Schichtaufbau Leitwert 2D : 1,021481 W/mK

U 1: 0,36918 W/m²K U 2: 0,21237 W/m²K

Länge 1: 1980 mm Länge 2: 1900 mm

U * l : 0,73097640 W/mK + U * l : 0,40350300 W/mK = Ges. U * l : 1,13447940 W/mK

Ψ - Wert: -0,112998 W/mK

OK

You can save these calculations and create a screenshot using the camera icon.

Chi-Wert-Berechnung [WK ⁻¹]			
Abmessung	per Hand	Antherm	
außen	$2,427679 - 1,021481 \cdot 1,92 - 0,999176 \cdot 1,98 - 0,723293 \cdot 1,90 +$ $+ 0,36918 \cdot 3,8016 + 0,21237 \cdot 3,648 + 0,19079 \cdot 3,762 = 0,00976$	0,00976	✓
innen	$2,427679 - 1,505 \cdot (1,021481 + 0,999176 + 0,723293) +$ $2,265 \cdot (0,36918 + 0,21237 + 0,19079) = 0,04738$	0,04742	✓

Inhomogenous Layer Calculator (Tool)

The Inhomogenous Layer Calculator can be used to compute the features of a thermally equivalent homogenous layer. Possible applications include brick masonry and wooden walls. On opening the Inhomogenous Layer Calculator first the Solver is called in case the Thermal Coupling Coefficients have not been calculated yet. As soon as that has been done, the width, thickness and (in 3D models) height of the largest element of the model are automatically inserted in the form. If due to the modelling this element does not have the measures relevant for the computation, the user can also insert these values manually. With manual input, the report does not contain annotations to the graphic as otherwise they might not fit to the model.

After clicking the button "Calculate" the report can be generated by clicking the button "Report".

Symbol	Description	Value	Unit
b	Layer Width	250	mm
d	Layer Thickness	200	mm
h	Layer Height	1000	mm
ρ	Mass Density		kg / m ³

Calculate

λ _{equ}	Equivalent Thermal Conductance		W / (m * K)
c _{equ}	Equivalent Thermal Capacity		kJ / (kg * K)
μ _{equ}	Equivalent Vapour Diffusion Resistance		mg / (m * Pa * h)
R	Transmission resistance		m ² * K / W
x (chi)	Areal heat capacity		kJ / (m ² * K)
m _{W,B,A}	Areal storage-effective mass		kg / m ²

Report

The following inputs are relevant:

- Width, Thickness and Height of the input model
- Mass Density of the input model (only relevant for the computation of the equivalent thermal capacity)

The following material attributes are calculated and displayed:

- Equivalent Thermal Conductance of the homogenous replacement model
- Equivalent Thermal Capacity of the homogenous replacement model (requires to enter the density and the availability of the Harmonic or Transient option)
- Equivalent Vapour Diffusion Resistance of the homogenous replacement model (requires Vapour option)

Furthermore, the following thermal parameters are computed and displayed:

- Transmission Resistance
- Areal heat capacity
- Areal storage-effective mass

Air Cavity Calculator (Tool)

The Air Cavity Calculator window is used as a tool for rapid calculation of equivalent heat conductivity of small air cavities and air gaps based on the input of their dimensions and heat stream direction.

To calculate equivalent heat conductivity (λ) the application uses formulas defining calculation of heat transfer resistance of air spaces described in EN ISO 6946:2005 Appendix B - please be aware of limitations related to its application (e.g. small, closed air cavities). Calculation regulation are geared to European norms EN ISO 6946 and EN ISO 10077-2 and cover two dimensional calculations.

This tool can be started from the [menu Tools](#).

The screenshot shows the 'Air Cavity Calculator' window with the following settings:

- Tab: Rectangular
- Heat Stream direction: Horizontal
- dT: 5.00 K
- Size in stream direction: 100.00 mm
- perpendicular to str. dir.: 10.00 mm
- Buttons: 5 K (EN 6946), 10 K (EN 10077)
- Equiv. Resistance: 0.2891 m²K/W
- Equiv. Lambda: 0.346 W/mK
- Buttons: Pick selected elements, Flip thickness/width

Heat Stream direction	Definition of heat stream direction: <ul style="list-style-type: none"> • Horizontal (default) • Upwards • Downwards
Size in stream direction	The size of the cavity measured in the heat stream direction (thickness). <u>Remark:</u> Thickness and width can be swapped via a context menu.
perpendicular to str. dir.	The size of the cavity perpendicular to the stream direction (width) <u>Remark:</u> Thickness and width can be swapped via a context menu.
Temperature difference dT	The difference of temperature between cavity boundaries in the heat stream direction. <u>Remark:</u> According to normative regulations one shall set the temperature difference, if exact value is not known, accordingly. Buttons "5K (EN 6946)" and "10K (EN 10077)" set this value to one of magnitudes required in these respective regulations.
Equiv. Resistance	Equivalent heat transfer resistance of particular air cavity
Equiv. Lambda	Equivalent heat conductivity of particular air cavity
Result	Calculation result is replaced by the actual value. A double click on this button results in an assignment of the calculation result to currently selected elements of element list. <u>Remark:</u> The button is also used as a source of a <i>drag-drop operation</i> of material property of a equivalent material.

Remark: An air cavity will consist of only one element rarely; usually an air cavity is described by a group of several elements.

For the very usual case of **non rectangular air cavities** normative regulations define following procedure to account for:

An equivalent conductivity is to be calculated for a fictive rectangular cavity. Such fictive cavity shall be constructed as follows:

- create a rectangle surrounding the cavity of interest.
- the area of a fictive cavity rectangle shall be equal to the area original (non rectangular) cavity.

The screenshot shows the 'Air Cavity Calculator' window with the following settings:

- Tab: Non-Rectangular
- Heat Stream direction: Horizontal
- dT: 5.00 K
- Size in stream direction: 100.00 mm
- perpendicular to str. dir.: 10.00 mm
- Buttons: 5 K (EN 6946), 10 K (EN 10077)
- Area of bound Airspace: 1000 mm²
- Equiv. Lambda: 0.346 W/mK
- Buttons: Pick selected elements, Flip thickness/width

- the ratio of width to the height of the fictive rectangle cavity shall be the same as of the rectangle surrounding the original.

The conductivity calculated for such fictive rectangular cavity is the used instead for the (non rectangular) original air cavity.

Heat Stream direction	Definition of heat stream direction: <ul style="list-style-type: none"> • Horizontal (default) • Upwards • Downwards
Thickness (stream dir.)	The thickness of the rectangle surrounding the cavity measured in the heat stream direction. <u>Remark:</u> Thickness and width can be swapped via a context menu.
Width	The width of the rectangle surrounding the cavity (perpendicular to the thickness) <u>Remark:</u> Thickness and width can be swapped via a context menu.
Area	The actual are of the real (non rectangular) air cavity
Temperature difference dT	The difference of temperature between cavity boundaries in the heat stream direction.
Equiv. Lambda	Equivalent heat conductivity of particular air cavity
Result	Calculation result is replaced by the actual value. A double click on this button results in an assignment of the calculation result to currently selected elements of element list. <u>Remark:</u> The button is also used a a source of a <i>drag-drop operation</i> of material property of a equivalent material.

Remark: For this procedure the most sophisticated it is to calculate the real are of the non rectangular cavity itself.

Air cavity dimension picked from element selection

This air cavity calculator provides a feature allowing automatic acquisition of area and dimension values from current [element selection](#). [Already](#) selected temperature difference and stream direction are not affected. Only dimensions of the rectangle surrounding selected element(s) and the sum of its/their respective area(s) are automatically calculated and transferred into calculator.

Possibly one will have to swap the thickness and width manually (via the context menu or respective button) to adjust to the fact, that the thickness must be specified parallel to the assumed stream direction!

Applying the result onto selected elements

A double click onto the button "Result" will apply the calculated result of equivalent conductance onto all material elements currently selected within the element list.

For the material's name a text "*Cavity dir t=..w=..(A=...) dT=...*" is used.

Remark: The result (equivalent lambda) can be drag-dropped from the button "Result" directly onto an element within [element editor](#) or onto elements ([Element list](#), [Elements 2D](#)) or event into [material list](#).

As an alternative, to assign the result (an equivalent lambda), one can use the clipboard also: Click onto the result field "Equiv. Lambda" with the right mouse button and select "Copy". Then you are able to paste the resulting value into the [lambda input field of the element editor](#) or [material editor within the material list](#) (by pasting it from the clipboard there).

Remark: The exact direction of stream flow is known in advance within multi dimensional calculations. This means, that the requirement to decide the correct choice of the "width" and "thickness" for the cavity cannot be fulfilled easily. It is user's decision to estimate the right heat stream direction by some heuristics based on his experience („along the x- or y-axis“).

Note: You shall switch between input fields by using the TAB-key. Using that key confirms the data entry. On the other hand it provides a good alternative compared to permanent mouse use.

Condensing Humidity (dew point) Calculator (Tool)

The Condensing Humidity Calculator (Dew Point Calculator) is used for rapid calculation of the [surface dew point](#) (i.e. relative humidity above which there is condensation probability on the surface) resulting from the input of surface and space air temperatures.

This tool can be started from the [menu Tools](#).

Space Temperature	The temperature of the space air.
Surface Temperature	The temperature of the surface under consideration
Condensing Humidity	The resulting dew point (maximum safe relative air humidity)
Mould Growth Humidity	The resulting maximum safe relative air humidity to avoid mould growth
Corroding Humidity	The resulting maximum safe relative air humidity to avoid corrosion

Condensing Humidity Calculator

Space Temperature: 24.00 °C

Surface Temperature: 5.00 °C

Condensing Humidity: 29.24 %

Mould Growth Hum.: 23.39 %

Corroding Humidity: 17.54 %

The formulas used for this dew point calculation are defined in the by formulas of partial pressure as shown in the EN ISO 13788:2002.

Note: You shall switch between input fields by using the TAB-key. Using that key confirms the data entry. On the other hand it provides a good alternative compared to permanent mouse use.

Additional remarks

- The standalone version of the tool can be invoked with an argument of "en" or "de" to work in the language different from the operating system default.

See also: [Result report](#), [Boundary conditions window](#), [Remarks to dew point](#)

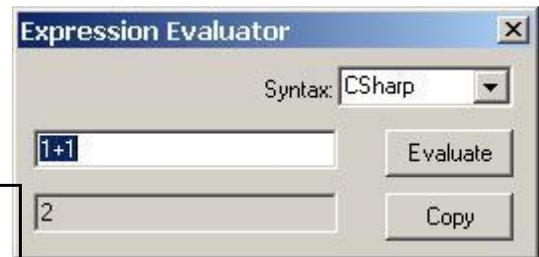
See also: [Element Editor](#), [Materials window](#), [Element selection window](#), [Elements 2D window](#)

Expression Evaluator (Tool)

This expression calculator is used as a tool for rapid evaluation of simple expressions, like $1/25$ or $\text{Math.Sin}(3.14)$. It can be helpful for calculating values required as [input of boundary conditions](#) or [input of material- or surface properties](#).

This tool can be started from the [menu Tools](#).

Syntax	For more complex expression one can select the syntax defined by programming languages like C# (C-Sharp) or VB (Visual Basic) from the .NET Family.
Input field	Input field for the entry of an expression to be evaluated.
Result field	Displays the result of evaluated expression.
Evaluate	Requests the evaluation of the expression shown in the input field.
Copy	Copies the contents of the result field into the clipboard making it available for the paste operation within other windows and its input fields.



Expression entered into the input field must conform to the syntax of the programming language chosen. To receive the result R of an expression E ($R = E$) one shall enter the expression term E into the input field.

All functions available in the respective programming language can be used also - especially mathematical functions like Math.Sin or Math.Log etc.

Further (non numerical) expressions can be used here also, but describing it is beyond the scope of this application.

Examples

Expression (C#)	Result	Comment
$1/15$	0	The result is 0 because integral types only are used in the expression
$1.0/15$	0,0666666666666667	The result in double precision
$1./15$	#Error!	Error, because floating point numbers are to be entered with at least one digit after the decimal point (C#)
$1f/15$	0,06666667	The result is single precision, due to the use of the suffix "f".
$\text{Math.Sin}(3.14)$	0,00159265291648683	Using the Math library
$\text{Math.Sin}(\text{Math.PI})$	1,22460635382238E-16	Using constants from the Math library
$\text{Math.Log}_{10}(10)$	1	
$\text{Math.Log}(\text{Math.E})$	1	
$33.3*(\text{Math.Log}(7)+1)$	98,0988079635419	Using expressions in braces

Remark: If the expression entered leads to an error there will be a pop-up window revealed explaining the cause.

Remark: Please be aware of the fact, that numeric constants must conform to the syntax of the programming language, thus the decimal comma (for example) might differ from the current [localisation of the system](#) as shown here above.

Remark: Detailed syntax descriptions for expressions:

- [C# Language Specification, Expressions](#)
- [Visual Basic Language Specification, Operator Expressions](#)

Note: You shall switch between input fields by using the TAB-key. Using that key confirms the data entry. On the other hand it provides a good alternative compared to permanent mouse use.

See also: [Boundary conditions window](#), [Element Editor](#)

Evaluation windows (Results windows)

Documentation of [input data](#) is offered as reports within following windows:

- [Data-Entry report](#)
- [Modelling report](#)

Application results are offered in following windows:

- [Solver](#)
- [Coupling Coefficients \(Leitwert\) report](#)
- [Psi-Value Determination](#) (in 2D projects)
- [Results report](#)
- [Results 3D](#) (the visualization of thermal or vapour diffusion, [temporal animations](#) too)
- [Timelines](#) (charting of temporal results und time dependant boundary conditions)

Evaluation parameters required for the calculation of specific results are input within following windows:

- [Boundary conditions](#) (and [Periodic / Harmonic Data Editor](#) for transient case)
- [Probe points](#)
- [Isometries](#)
- [Results3D parameter](#)

If some (additional) computation is required for respective results to be shown in a requested evaluation window, it will be initiated automatically.

See also: [Editing windows \(Input windows\)](#), [Dialog windows](#), [Main menu](#), [Results and Evaluations](#), [Main window](#), [Toolbar of report windows](#)

Data-Entry report

The Data-Entry report is used for detailed documentation of all data entered while modelling a building component. The window can be opened from the [Main menu](#) via [→Evaluation & Reports Report builder or Modelling Report→](#) and via [Data Input & Entry→Status Report](#)

Kornicki, Dienstleistungen in EDV und IT
A-1230 Wien, Othellogasse 1/RH 8/2
Tel./Fax. +43-1-6157099
email: tkornicki@chello.at



28.05.2020
AnTherm
V.10.137.1
(c) Kornicki, all rights reserved

Beispiel zum dreidimensional ablaufenden Waerme- und Dampftransport (Diffusion)
Anschlussdetail im Bereich des Flachdachs eines Hallenbads
Trennwand zwischen Wohnraum und Hallenbad: zeichalige Backsteinwand mit 5 cm Kerndaemmung
Aussenwand des Hallenbads: 25 cm Stahlbeton, 15 cm Kerndaemmung, 12,5 cm Vormauerung
Berechnung des Waerme- und Dampftransport

Details of the construction's input data

No.	No.	C	X1	Y1	Z1	X2	Y2	Z2	DX	DY	DZ	T	Details
1											1000		ungestoert
1	1		-1000	-500	0	1500	1050	1000	2500	1550	1000	M	Stahlbeton
1	2		-1000	-500	0	1500	0	1000	2500	500	1000	S	Hallenbad / Hallenbad
1	3		500	0	0	1500	750	1000	1000	750	1000	S	Hallenbad / Hallenbad
1	4		0	250	0	150	2150	1000	150	1900	1000	M	Backstein innen
1	5		150	250	0	200	2150	1000	50	1900	1000	M	Daemmstoff
1	6		-1000	250	0	0	2150	1000	1000	1900	1000	S	Wohnraum / Wohnraum
1	7		200	1050	0	325	1150	1000	125	100	1000	M	Schaumglas
1	8		200	1150	0	325	2150	1000	125	1000	1000	M	Backstein aussen
1	9		325	1050	0	1500	2150	1000	1175	1100	1000	S	aussen / aussen
1	10		325	1050	0	1500	1150	1000	1175	100	1000	M	PS-Hartschaum
1	11		325	1150	0	1500	1155	1000	1175	5	1000	M	Bikutop
1	12		200	1048	0	1500	1052	1000	1300	4	1000	M	Sopravap
1	13		150	250	0	150,2	2150	1000	0,2	1900	1000	M	PVC-Folie
1	14		325	1150	0	330	1250	1000	5	100	1000	M	Bikutop
1	15		330	1155	0	1500	1245	1000	1170	90	1000	M	Kies
1	16		-1000	250	0	0	270	1000	1000	20	1000	M	Sandausgleich
1	17		-1000	270	0	0	290	1000	1000	20	1000	M	Trittschalldaemplatte
1	18		-1000	290	0	0	340	1000	1000	50	1000	M	Zementestrich
1	19		-1000	340	0	0	360	1000	1000	20	1000	M	Parkett

Remark: Data of vapour diffusion resistance μ will be also reported if entered.

A report can be:

- viewed on the screen
- saved as PDF, XLS, DOC or RTF file
- printed onto a printer

See [Toolbar of a Report window](#)

Remark: The content of this report will be updated after any change to input data immediately. This might result in slight delays during processing of the input. Therefore it is advisable to keep that report window closed during the input.

See also: [Evaluation windows](#), [Evaluation and Results](#), [Toolbar of a Report window](#)

Modelling report

The Modelling report is used for summarizing documentation of a building components data relevant for the simulation. The window can be reached from the [Main menu](#) via [→Evaluation & Reports→Modelling report](#) and via [Evaluation & Reports→Report builder](#)

Kornicki, Dienstleistungen in EDV und IT
A-1230 Wien, Othellogasse 1/RH 8/2
Tel./Fax: +43-1-6157099
email: tkornicki@chello.at



AnTherm
V.10.137.1
(c) Kornicki, all rights reserved

28.05.2020

Beispiel zum dreidimensional ablaufenden Wärme- und Dampfsreport (Diffusion)
Anschlussdetail im Bereich des Flachdachs eines Hallenbads
Trennwand zwischen Wohnraum und Hallenbad: zweifelhige Backsteinwand mit 5 cm Kerndaemmung
Aussenwand des Hallenbads: 25 cm Stahlbeton, 15 cm Kerndaemmung, 12,5 cm Vermauerung
Berechnung des Wärme- und Dampfsreports

Layered constructs and U-Value calculations

aussen <-> Hallenbad @ BottomLeft: (-1000, -500, 1475,4) x (-1000, -500, 1000)

Material / Surface	λ [W/(m·K)]	d [mm]	R_s [m²KW]	α [W/m²K]	R [m²KW]
aussen/aussen			0,0400	25,0000	0,0400
Backstein aussen	0,6700	125,0000			0,1866
Daemmstoff	0,0400	100,0000			2,5000
PVC-Folie	0,2000	0,4000			0,0020
Stahlbeton	2,3000	250,0000			0,1087
Hallenbad/Hallenbad			0,2500	4,0000	0,2500
Σ		475,4000			U-Value: 0,3259 [W/m²K]

aussen <-> Wohnraum @ TopLeft: (-1000, 2150, 1475,4) x (-1000, 2150, 1100)

Material / Surface	λ [W/(m·K)]	d [mm]	R_s [m²KW]	α [W/m²K]	R [m²KW]
aussen/aussen			0,0400	25,0000	0,0400
Backstein aussen	0,6700	125,0000			0,1866
Daemmstoff	0,0400	100,0000			2,5000
PVC-Folie	0,2000	0,4000			0,0020
Backstein innen	0,4400	150,0000			0,3409
Wohnraum/Wohnraum			0,2500	4,0000	0,2500
Σ		375,4000			U-Value: 0,3013 [W/m²K]

aussen <-> Hallenbad @ BottomRight: (1500, -500, 1475,4) x (1500, -500, 1000)

Compared to the [Data-Entry report](#) only data relevant for the calculation is reported.

Resulting from the overlapping elements some materials might have no impact within calculation (being fully overlapped and thus discarded). Thus this report shall be used to document and to verify the input of spaces and power sources (boundary conditions), their names and transfer coefficients.

Remark: If there are power sources included in the model (after all overlapping of elements have been resolved) their resulting (active) volumes will be calculated and output also. This values might be helpful during the entry of [boundary conditions](#).

Remark: Data of vapour diffusion resistance μ will be also reported if entered.



The second part of the report lists **U-Values of characteristic layered constructs** at adiabatic boundaries of the model. These intermediary results are required for the calculation of [thermal bridge correction factor Psi or Chi](#).

Remark: The listing does only show U-Values of layered constructs between two spaces; Incomplete layered constructs will not be output. The algorithm searches at all six sides of the model bounding box for adiabatic cut-off planes (from outside towards model's interior, the first one adiabatic plane in each such direction). If a space surface is found at first in one of that directions, then there will be no adiabatic boundary reported in that direction. Adiabatic planes found by that process are then intersected in pairs at twelve edges. At these intersection lines U-Value profiles are generated - each such profile results in none, one or even many layered constructs space-to-space.

Remark: For U-Value profiles passing multiple layered constructs (e.g. multiple ceilings) there will be multiple U-Values reported for respective space pairs.

Solver window

The window *Solver* allows observing the progress of the simulation process.

The animation displayed at the top edge of the window signals that the application is currently calculating the solution. Further detailed messages reported by the computation will be shown within text window.

The simulation is executed in background completely (rasterizing, system of equations, optimal omega, calculation of base solutions, calculation of thermal coupling coefficients) without any further interaction.

If necessary the solution of vapour diffusion equations will be created also.

If solving transient, harmonic, periodic problem has been requested also, then respective system of harmonic equations will be set up for all chosen period lengths and solved too.

If there are [simulation results](#) already available, current input data will be compared to the one already saved. If there is no difference to the model calculated earlier the simulation step will be skipped. Calculation will start wherever:

- [Simulation results](#) are not available or are incomplete (partially solved)
- The model data (geometry, materials etc.) has been changed
- Different resulting discretization (changes to [fine grid parameters](#))
- More stringent [solver parameters](#) chosen (smaller termination condition, more iterations below termination condition, more final iterations).

Remark: The animation symbolically showing the progress of the computation (displayed at the top edge of the window) can be hidden via the context menu of that element (right mouse button). See also the [application setting "Animate Progress on Solver Form"](#).



Data integrity and messages

Relevant information about the progress or errors of the simulation process are shown within this window. Messages shown will not always report the reason for an error, albeit they shall be used as starting point for problem analysis and for further search for input error (e.g. $\lambda = 0$, no heat transport at the surface, only one space named, all spaces unnamed etc.)

Before actual computation is started the application examines the validity of input data. Following integrity checks are executed:

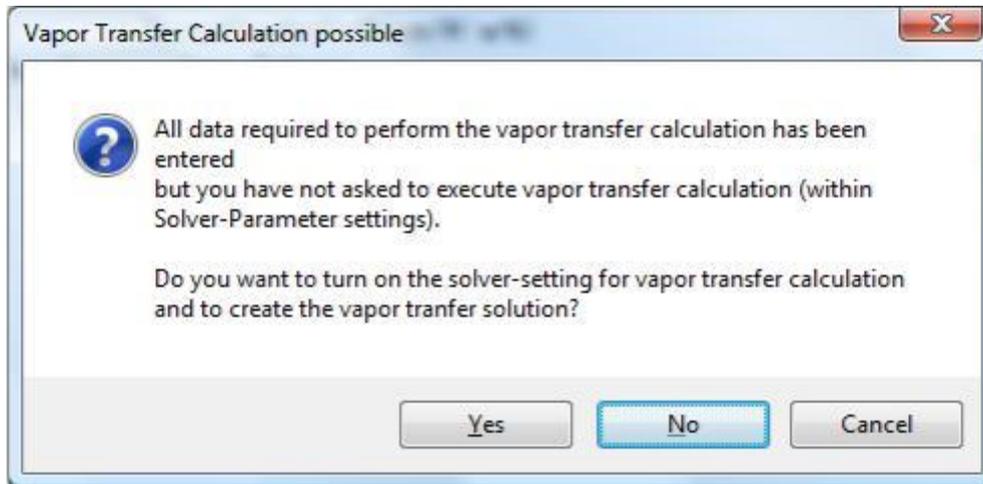
- the input must result in a 3D structure (even a 2D project will produce a 3D structure, a homogenous one in the Z axis -direction),
- heat conductance of all materials must not be zero,
- vapour diffusion resistance of all materials must not be zero (only relevant with active [VAPOUR-Option](#))
- volumetric heat capacities of all materials (product of mass density and heat capacity) must not be zero (only relevant with active HARMONIC/TRANSIENT option)
- transfer coefficients of all surfaces must not be zero,
- there must be at least two independent boundary conditions (different space- and/or power source names)
- there is at least one boundary condition of constant temperature (a space of defined air temperature - i.e. defined space name)
- names of power sources must differ from space names
- the resulting system of equations must contain at least two equations (i.e. at least two cells to calculate)
- availability of sufficient [license features](#)

If the resulting fine grid consists of 1.000.000 or more cells you will be shown a warning message (as set with the [application setting "Warn on excessive Equation Size"](#)). This provides the ability to stop the computation just before the equation system data, very large in such case, is being created and written to files (even the rasterizer tries to avoid

holes of the construction, which have no influence onto resulting equations, the size of the equation might be still above the number of cells reported).

If there is sufficient data provided for execution of vapour diffusion calculation, but the [Solver setting of vapour calculation](#) is turned off, the application will ask (when the VAPOUR license option if available) if this [setting of the solver](#) shall be turned on (see also: [Compute Vapour Transfer Solution inquiry](#)).

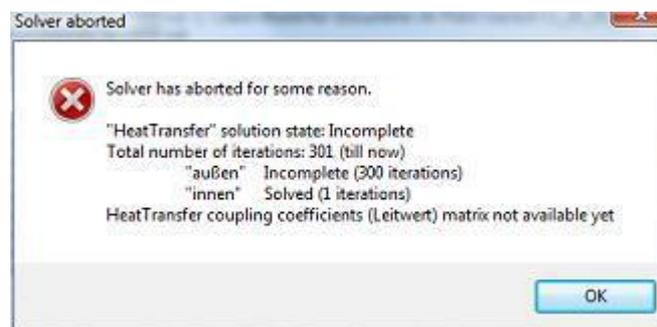
If there is sufficient data provided for execution of transient, harmonic, periodic calculation, but the [Solver setting of harmonic calculation](#) is turned off, the application will ask (when the HARMONIC/TRANSIENT license option if available) if this [setting of the solver](#) shall be turned on (see also: [Compute Transient Solution inquiry](#)).



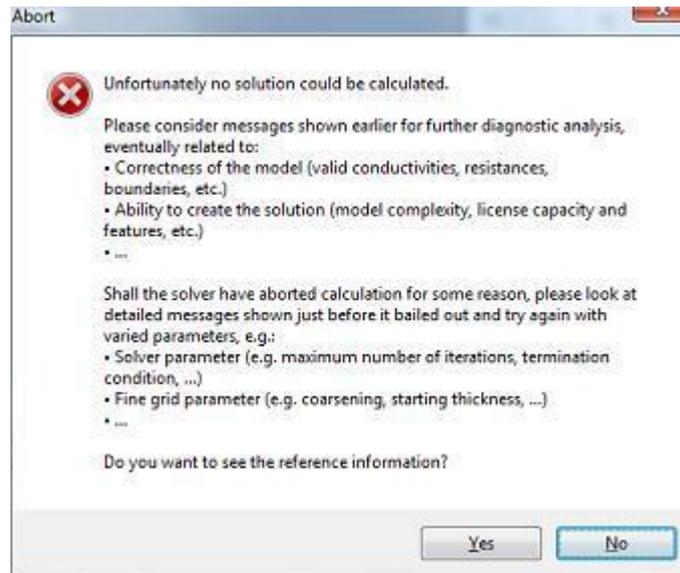
Stopping the calculation

Pressing the button *Stop* will initiate the abortion procedure of the solver. Terminating the solver will typically last 10 to 30 seconds.

Message: Solver aborted / Solver status



If the computation cannot be started or executed successfully the application will show a dialog window next to solver window providing further details and problem descriptions - prior to closing this window please analyse contents of solver window in the context of the message shown also.



Running Solver on multiple CPUs

Running solver on multiple CPUs enhances the calculation performance and thus (in most cases) reduces significantly the calculation time. [Multi CPU \(or Multicore\) behaviour of the solver is controlled by respective application settings](#) separately.

Important: Actual reduction of calculation time by execution of the solver on multiple processors results from various properties of employed hardware architecture (number of processors and processor cores, number and size of processor cache memories, cache speed, bandwidth to and from the RAM, Memory speed, bus speed and bus waiting rates, etc.). Some multicore systems might even show performance degradation if multiple cores simultaneously access shared resources.

It is advisable to compare the number of iterations executed per time unit (as displayed within the solver window).

Remark: This function requires a valid license feature [MULTICORE](#).

Remark: If the number of equations (independent variables) to solve is less or equal the number of processors than each variable will be scheduled to one processor. Due to synchronization required on every iteration poor performance may result for such small equations.

MULTICORE-option: *Speeding up computationally intensive jobs by distributing them on multiple processors or processor cores for parallel execution is only possible with an active MULTICORE-Option of the program.*



See also: [Evaluation windows](#), [Solver parameter](#), [Fine Grind parameter](#), [Rasterizing](#), [Application settings](#)

Coupling Coefficients and Precision report

Komicki, Dienstleistungen in EDV und IT
A-1230 Wien, Othellogasse 1/RH 8/2
Tel./Fax: +43-1-6157099
email: tkomicki@chello.at



29.05.2020
AnTherm
V.10.137.1
(c) Komicki, all rights reserved

Beispiel zum dreidimensional ablaufenden Waerme- und Dampftransport (Diffusion)
Anschlussdetail im Bereich des Flachdachs eines Hallenbads
Trennwand zwischen Wohnraum und Hallenbad: zweiseitige Backsteinwand mit 5 cm Kerndaemmung
Aussenwand des Hallenbads: 25 cm Stahlbeton, 15 cm Kerndaemmung, 12,5 cm Vorraeuerung
Berechnung des Waerme- und Dampftransports
Number of evaluated cells: 94502

Thermal Coupling Coefficients [W / K]

Space\Space	aussen	Hallenbad	Wohnraum
aussen		1,386223	1,292194
Hallenbad	1,386223		1,476608
Wohnraum	1,292194	1,476608	

Used Heat Transfer Coefficients

Space	Rs-Value (H)	Surface
aussen	Rs=0,04 m²K/W	aussen
Hallenbad	Rs=0,25 m²K/W	Hallenbad
Wohnraum	Rs=0,25 m²K/W	Wohnraum

Space	Rs-Value (T)	Surface
aussen	Rs=0,04 m²K/W	aussen
Hallenbad	Rs=0,25 m²K/W	Hallenbad
Wohnraum	Rs=0,25 m²K/W	Wohnraum

Precision information

Space	Close-up error [W / K]	Coeff. sum [W / K]	Relative close-up error
aussen	3,83693e-013	2,678417	1,43254e-013
Hallenbad	2,30616e-012	2,862831	8,05551e-013
Wohnraum	-2,68452e-012	2,768802	-9,69560e-013

The *Coupling Coefficients and Precision report* displays the matrix L_{ij} of [thermal coupling coefficients](#) (L^{2D} or L^{3D} , also called heat transfer coefficients H in some standards or Leitwert matrix, thermal conductance matrix) and respective [precision information](#) of the simulation. For a two dimensional model the matrix shows length related transmittance L^{2D} [$Wm^{-1}K^{-1}$]; for three dimensional construction the matrix displays thermal coupling coefficients L^{3D} **per se** [WK^{-1}]. Depending on which problems have been chosen for the Solver and on the number and type of boundary conditions following results are shown:

- Number of evaluated cells(size of the system of equations)

for the steady state (stationary) heat transport problem:

- Matrix of stationary, [steady state thermal coupling coefficients](#) for all space pairs
- Heat distribution factors for all power sources and spaces
- [Precision information](#) the steady state heat transfer problem

for the steady state vapour diffusion problem:

Precision information for the steady state vapour diffusion problem

for the dynamic,transient,harmonic,periodic heat transport problem,for any period length (and eventually higher harmonics):

- the matrix of [harmonic coupling coefficients](#) for all space pairs,
 - the [harmonic heat distribution factors](#) of each source to all spaces
- given as complex numbers and as amplitude/phase pairs.

If the number of matrix columns output shall overrun the page width then the output of overrunning matrix columns is continued in groups one below the other.

Steady state (stationary) thermal coupling coefficients

The report displays the matrix of thermal coupling coefficients for any pair of spaces (with 6 decimal places).

If N spaces are attached to the considered construction the NxN matrix will be displayed (without diagonal elements).

Theoretically the matrix shall be symmetrical (i.e. $L_{ij}=L_{ji}$), thus the output allows [precision consideration of results](#).

These values are used, for example, to calculate thermal bridge correction factors - for a 3D case the "point thermal transmittance" X (Chi) and for a 2D case the "linear thermal transmittance" Ψ (Psi).

See also: [Psi-Value Determination \(Calculate \$\Psi\$ -Value\)](#)

Remark: By multiplying the (steady state) thermal coupling coefficient L_{ij} by the difference of temperatures of respective space pairs $\Theta_i - \Theta_j$ one will receive the heat stream (or the length related heat stream in 2D case) between the two spaces transmitted through the modelled component.

Heat source Distribution Factors (steady state)

In the event heat sources are available also, the heat distribution factors of each source to all spaces are shown too.

Distribution factors [1]		
Powers.\Space	Room 0	Room 1
PS 0	0,111853	0,888147

If N spaces are attached to the considered construction then there will be N numbers shown for every heat source in the distribution table. The i -th ($i = 1, N$) column value of the distribution table shows the percentage of the heat provided by the particular heat source passing to the i -th space. The values of the distribution table are therefore from the range 0 to 1.

Because the steady state calculation does not cover the heat capacity storage, the sum of all distribution values must theoretically result in 1 (apart from minor rounding errors) allowing further [precision consideration of results](#).

Remark: By multiplying the distribution factor F_{kj} by its respective power density Φ_k of the heat source k and its volume V_k one will receive the heat stream (or the length related heat stream in 2D case) from the heat source k to the space j . (The volume of every heat source will be shown within [Modelling report](#) or [Results report](#)).

Transient (instationary, dynamic), harmonic, periodic thermal coupling coefficients

Provided the solution of a dynamic, transient, harmonic, periodic problem has been computed also, the matrix of periodic harmonic coupling coefficients will be output (with output of up to 4 decimal places).

The output is provided for each requested period (period length, in decreasing order; longest first) once as matrix of complex numbers and additionally as matrix of norm (amplitude) and argument (phase shift, time lag) values.

From that output the exterior harmonic coupling coefficient (between exterior and interior) can be identified for example.

Diagonal elements (contrary to the steady state case these are significant and output here) can be used for the calculation of effective heat capacities.

Theoretically, as for the steady state case, the matrix shall be symmetrical, thus the output allows precision considerations of results also.

Kornicki, Dienstleistungen in EDV und IT A-1230 Wien, Othellogasse 1/RH 8/2 Tel./Fax. +43-1-6157099 email: tkornicki@chello.at		29.05.2020	
		 AnTherm V.10.137.1 (c) Kornicki, all rights reserved	
Number of evaluated cells: 104090			
Thermal Coupling Coefficients [W / K]			
Space\Space	Außen	Innen	
Außen		1,746472	
Innen	1,746472		

Harmonic Thermal Conductance for period: 86400 s Day

Space/Space	Außen		Innen	
	Re	Im	Re	Im
Außen	-6,3238	-5,6256	-0,1332	-0,0678
Innen	-0,1332	-0,0678	-3,6580	-1,5798

Space/Space	Außen		Innen	
	Amplitude [W/K]	Phase [hours]	Amplitude [W/K]	Phase [hours]
Außen	8,4639	-9,2229	0,1494	-10,2009
Innen	0,1495	-10,2014	3,9846	-10,4428

In the event heat sources are available also, the [harmonic heat distribution factors](#) of each source to all spaces are shown too.

Vapour diffusion, hygric coupling coefficients

The report will also show *hygric coupling coefficients* (mg/Pa*h) ("Matrix of hygric coupling") if there are results of vapour diffusion calculation available too.



Vapour Diffusion Coupling Coefficients [mg / m*Pa*h]

Space\Space	aussen	Hallenbad	Wohnraum
aussen		0,002463	0,164889
Hallenbad	0,002463		0,075282
Wohnraum	0,164889	0,075282	

Precision information

An immediate indicator of the precision of the solutions is the matrix of thermal coupling coefficients itself - if it is not reasonably symmetrical, then further calculation might be necessary. Along with the output of coupling coefficients the **necessary information about the precision** of these results is output.

A theoretically exact method of calculation would necessarily satisfy the condition, that the sum of heat entering a component exactly equals the sum leaving the same - for any set of basic solutions. The inherent nature of a numeric method, however, will always result in a marginal difference in the evaluated energy balance. This is referred to here as the *close-up error* of base solution. Dividing the close-up error by the sum of the coefficients associated with the respective base solution (space) provides a measure of the precision of calculation.

Precision information

Space	Close-up error [W / m*K]	Coeff. sum [W / m*K]	Relative close-up error
aussen	5,12985e-007	0,892497	5,74775e-007
Hallenbad	6,23260e-007	2,315464	2,69173e-007
Wohnraum	-1,13624e-006	2,460200	-4,61851e-007

The relative close-up error shall never exceed 10^{-4} considering precise solution (see EN ISO 10211:2008).

The [value of the Relative Close-Up Error limit](#) (default 10^{-4}) can be adjusted within application settings:

- If the relative close-up error exceeds the half of that limit the line will be marked with (*) - i.e. "just at minimum precision".
- If the relative close-up error exceeds that limit (**) are shown.

A warning message "(*) *Warning: The precision criterion concerning the magnitude of relative close-up errors is not fulfilled*" will be shown below the precision data listing if the precision criterion is not satisfied by any of base solution (the display of this message can be turned off by the application setting "[Relative Close-Up Error - Warn if above](#)").

Note: If the relative close-up error exceeds 10^{-4} (or [the limit otherwise set](#)) a continuation of calculation based on more stringent [parameters for more precise solution](#) should be considered before proceeding with further evaluation. Eventually modified (finer or coarser) raster discretisation ([gridding](#)) might be required also.

If the solution of vapour diffusion has been also computed, the associated precision information is also displayed following the results of thermal heat calculation.



Precision information (Vapour Diffusion calculation)

Space	Close-up error [mg / m ³ Pa ³ h]	Total sum [mg / m ³ Pa ³ h]	Relative close-up error
aussen	2,60264e-009	1,67352e-001	1,55519e-008
Hallenbad	1,94416e-009	7,77448e-002	2,50069e-008
Wohnraum	-4,54678e-009	2,40171e-001	-1,89315e-008

Precision indicators, i.e. (*), (**) and a warning message, are shown for the vapour base solutions - based on same [precision criteria](#) (see above).

A report can be:

- viewed on the screen
- saved as [PDF, XLS, DOC or RTF](#) file
- printed onto a printe
- searched by keywords

See [Toolbar of a Report window](#)

See also: [Evaluation windows](#), [Evaluation and Results](#), [Toolbar of a Report window](#), [Results 3D window](#), [Boundary Conditions window](#), [Please Wait window](#), [Psi-Value Determination \(Calculate Ψ-Value\)](#), [Linear and Point Transmittance](#), EN ISO 10211

Psi-Value Determination (Ψ -Value Calculation)

The Psi-Value Determination Form is used for automatic calculation of the **linear thermal transmittance Psi** of the (linear) thermal heat bridge (correction factors Psi (Ψ , L_{Ψ} ,) for the currently analysed **two dimensional (2D) building component** according to the EN ISO 10211 (or EN ISO 10077).

Note: The Psi value is a computational value that corrects the error of the one-dimensional computation. The Psi value must not be assigned a physical significance.

Most typical 2D cases will be calculated without any additional user intervention. The automation is supported by:

- calculation of the **thermal coupling coefficient L2D** (as calculated within by the steady state, **two dimensional thermal heat bridge simulation**),
- automatic identification of **characteristic U-Values at adiabatic boundaries of the model (also shown within the Modelling Report)** and
- determination of respective lengths (exterior- or interior based) resulting from the **Reference Point** which are determined automatically if possible.

Some limitations of that automation depend on the particular geometry and can be controlled by the user.

Remark: Psi-Value Determination Form will show disabled if neither automatic nor partly automatic determination is feasible for the particular model. The message related to such situation will be displayed also. Manual arbitrary calculation of Psi-Value is then supported by the **Psi-Value Calculator tool** only.

Remark: The model is interpreted as two dimensional if it is homogenous in the Z direction (the grid contains only one layer in Z).

Space to assume -Exterior -Interior	<p>Select the space name which is to be assumed exterior at first. The choice of the value of the thermal coupling coefficient L^{2D}, the automatic determination of the Reference Point and the selection of characteristic U-Value Profiles at adiabatic boundaries all depend on that input.</p> <p>By default AnTherm will pick the name of the first space (as sorted in alphabetic order) which in particular is assigned the lowest temperature during later evaluations.</p>
Coupling Coefficient L^{2D}	<p>The thermal coupling coefficient specific to the component under consideration (as calculated and reported within the Coupling Coefficients report, for example).</p> <p>As soon as simulation results of a two dimension calculation are available or created within AnTherm the Psi-Value Determination Form will pick the value of the coupling coefficient between the two spaces (exterior and interior) automatically. The value transferred is the median of the two corresponding matrix elements divided by the thickness in Z (in 2D it is 1 meter) and rounded according to the close-up error.</p>

U-Value Profiles at adiabatic cut-off planes	<p>Turn that switch on to request AnTherm to pick the profiles automatically at adiabatic boundaries. Available U-Value Profiles are then shown in the two choice boxes displayed just below the switch for further selection. If turned off you will have to provide U-Values manually (either by the value input or as calculated with a U-Value Calculator)</p> <p>By default AnTherm attempts to automatically find appropriate profiles. <u>Remark:</u> If profiles cannot be found this control element will be shown inactive (greyed).</p>
U-Value Profile selection	<p>Characteristic profiles at adiabatic cut-off planes found automatically are available for selection. <u>Remark:</u> AnTherm will offer only profiles connected to the exterior or interior spaces.</p> <p>The layered construct at the profile chosen is automatically created and passed to the respective U-Value Calculator in the fields shown below, thus replacing the earlier entries therein.</p> <p>By default AnTherm will automatically pick the first two orthogonal profiles from the list available.</p>
Layered construct	<p>The button provides the means of viewing (or entering) a layered construct for the calculation of its characteristic U-Value.</p> <p><u>Remark:</u> The layered construct editor (the U-Value Calculator) will block changes (editing buttons and menus will be grayed) if the construct is provided from the profiles identified by AnTherm. If you wish to override the construct by changing it you must uncheck the "U-Value Profiles at adiabatic cut-off planes" switch above.</p> <p><u>Remark:</u> If specific U-Value is calculated from a layered construct entered within a U-Value calculator (or from the characteristic profile selected), the checkmark left to the button "Layered Construct" is marked checked. To override the U-Value manually uncheck the checkmark left to the button "Layered Construct".</p>
U-Value	<p>The U-Value specific the the components part concerned (as calculated with a U-Value Calculator, for example).</p> <p><u>Remark:</u> If specific U-Value is provided by a layered construct entered within a U-Value calculator, the checkmark left to the button "Layered Construct" is marked checked and this field is marked read-only. To override the value manually uncheck the checkmark left to the button "Layered Construct".</p>
Length	<p>The length to be used for the respective U-Value above.</p> <p><u>Remark:</u> If respective lengths are determined automatically from the Reference Point, the respective fields are shown read-only.</p>
Determine Lengths by Reference Point	<p>Lengths shall be determined from the specific Reference Point found automatically for exterior or interior dimensions or given by manual entry.</p> <p><u>Remark:</u> If respective lengths cannot be determined from any Reference Point that choice box and controls below it are shown disabled..</p>
Reference Point at -exterior -interior -manual	<p>The Reference Point shall be found automatically at exterior or interior dimensions or specified by manual entry.</p> <p><u>Remark:</u> If the respective reference point cannot be determined at some specific automation, that choices are shown disabled.</p> <p>See the explanation graphics below.</p>
U * l	The result of the multiplication of respective values of U and l.
Total U * l	The sum of respective U * l multiplications.
Psi-Value (Ψ)	The resulting Psi-Value (Ψ , L_{Ψ} ,); the linear thermal transmittance

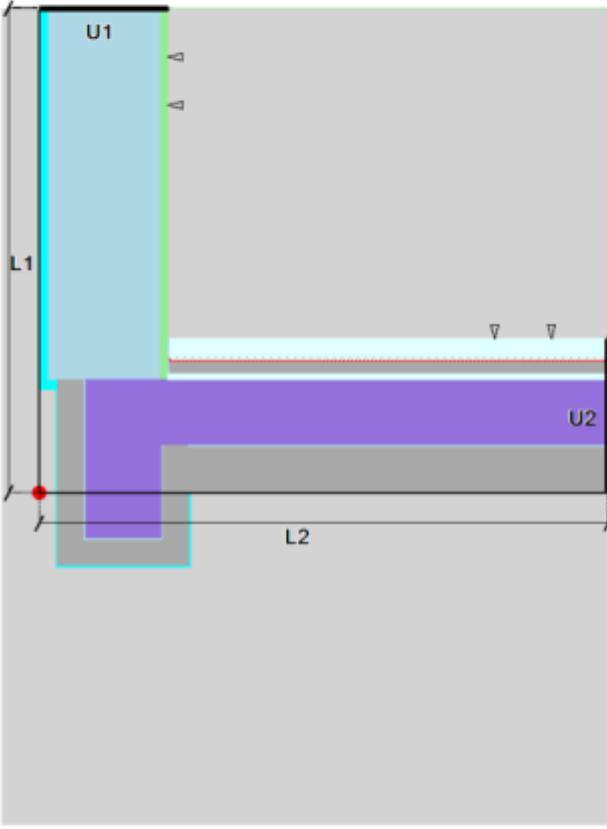
Psi-Value Computation
Page 1

Kornicki, Dienstleistungen in EDV und IT
 A-1230 Wien, Othellogasse 1/RH 8/2
 Tel./Fax. +43-1-6157099
 email: tkornicki@chello.at

7/6/2016
 AnTherm V.8
 (c) Kornicki, all rights reserved

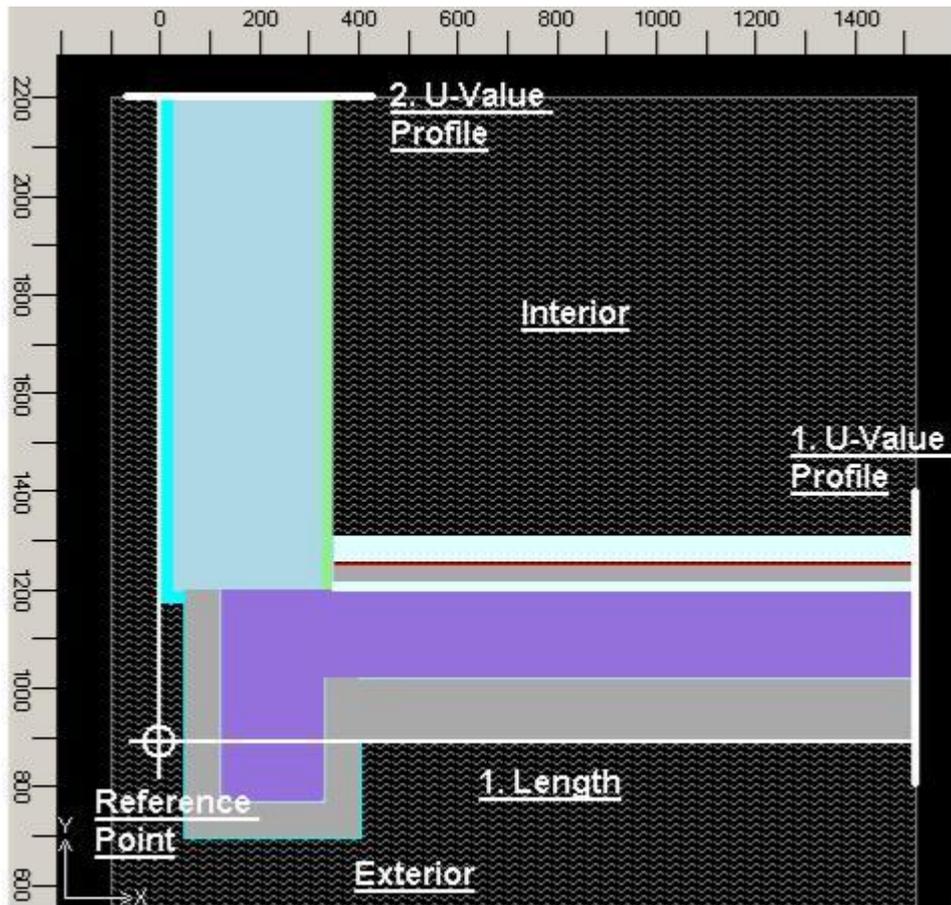
TUTORIAL - 2-dimensional:
Reinforced concrete slab with masonry bearing wall on girder over

Psi-Value Computation



$L_{2D} = 0,9206 \text{ W/mK}$	
$U_1 = 0,4772 \text{ W/m}^2\text{K}$	$L_1 = 1,305 \text{ m}$
$U_2 = 0,2464 \text{ W/m}^2\text{K}$	$L_2 = 1,525 \text{ m}$
$\Psi = L_{2D} - U_1 L_1 - U_2 L_2 = -0,0779 \text{ W/mK}$	

AnTherm - Heat & Vapour Transfer - Thermal Bridges- Walter,Udo,Klaus,Tomasz
 Copyright (c) 2003-2016 M.Kornicki Dienstleistungen in EDV & IT <http://www.kornicki.eu>



Calculating U-Values with U-Value Calculator

To calculate respective U-Value with an integrated [U-Value Calculator](#), press the button "Layered Construct", build or load the layered construct within the [U-Value Calculator](#) which is shown thereafter and confirm the calculated result by pressing the Ok-button therein. The calculated U-Value of a layered construct is shown in the U-Value field which is then marked read-only. The checkmark left to the button "Layered Construct" confirms, that the value shown is based on respective layered construct.

Layered construct
U-Value: 0.434711237 W/m²K

The actual data of a layered construct can be changed at any time by pressing the button "Layered Construct" again and editing the data within the U-Value Calculator shown thereafter.

Remark: The actual data of a layered construct is retained until the form is closed. To save the data of each respective layered construct use the "Save" function within the [U-Value Calculator](#).

Remark: You can drag & drop any layered construct (as it is available from a [U-Value Calculator](#) if it is run separately) onto the one of U-Value / Length blocks in this window also.

Remark: To override the value manually uncheck the checkmark left to the button "Layered Construct".

Remark: If no data of a layered construct has been applied yet, the checkmark left to the button "Layered Construct" is shown greyed.

Thermal Coupling Coefficient L^{2D} obtained from AnTherm

[AnTherm's "Coupling Coefficients Report"](#) provides the matrix of thermal coupling coefficients resulting from steady state simulation of respective modelled construction. As AnTherm displays both, the interior and exterior calculated values for precision validation purposes, the mean value of both is used.

Important: U-Values and lengths used for the Psi-Value calculation must correspond to the thermal coupling coefficient of the modelled component in the manner defined within EN ISO 10211 and EN ISO 13789.

As soon as simulation results of a two dimension calculation are available or created in AnTherm the Psi-Value Calculator will pick the value of the thermal coupling coefficient between the two selected spaces automatically. Remark: The model is interpreted as two dimensional if it is homogenous in the Z direction (the grid contains only one layer in Z).

Remark: The value transferred is calculated as the mean value of the two matrix elements which apply and divided by the thickness in the Z direction (in 2D it is 1 Meter) and then rounded according to the current close-up error.

U-Values obtained automatically by AnTherm

[AnTherm's "Modelling Report"](#) shows U-Values for all layered constructs at adiabatic cut-off planes of the modelled component.

Important: U-Values and lengths used for the Psi-Value calculation must correspond to the thermal coupling coefficient of the modelled component in the manner defined within EN ISO 10211 and EN ISO 13789.

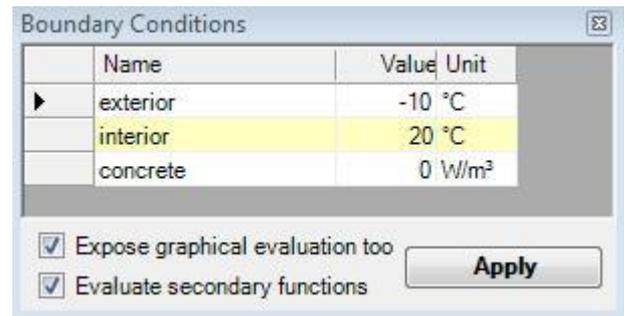
See also: [Coupling Coefficients and Precision report](#), [Modelling report](#), [Psi-Value Calculator \(Tool\)](#), [U-Value Calculator \(Tool\)](#), [Project types](#), [Linear and Point Transmittance](#), [On calculation of \$\Psi\$ -values for building constructions in connection with ground](#), EN ISO 14683, EN ISO 13789

Boundary Conditions window

The window *Boundary Conditions* enlists all spaces of which the user sets temperatures and all power sources of which power density can be entered (the boundary conditions). For dynamic, transient problem solution the window mutates to allow [input of time dependant quantities](#).

Temperatures of spaces are initially preset to standard values:

- first space in the list (sorted alphabetically) is set to airspace temperature of -10°C
- all other spaces are set to the temperature of $+20^{\circ}\text{C}$.



Remark: These standard values can be changed within [9,5](#)

Spaces are enlisted alphabetically. Knowing this, by consequently naming spaces you can target proper automatic assignment of preset values.:

- *Exterior* precedes *Interior*
- *Space 00* precedes *Space 03* and *Space 10*
- *Space 11* precedes *Space 1*.

If the component contains power sources (heat sources or sinks) also, their names will be offered here for the input of their respective power density [W/m^3]. Initially power sources are set to have no power (equal $0 \text{ W}/\text{m}^3$).

Remark: If there is solution of vapour calculation available too, the input of relative humidity of air space, required for vapour evaluations of partial pressure, is also offered in additional dedicated column:

	Name	Value	Unit	Rel.Humidity	Part.Pressure
▶	Room 0	-12	$^{\circ}\text{C}$	80 %	1.734 hPa
	Room 1	20	$^{\circ}\text{C}$	53 %	12.386 hPa



Relative humidity of air spaces is initially preset as follows:

- first space is assigned the relative air humidity of 80%
- all further spaces are assigned relative air humidity of 53%.

Remark: These standard values can be changed within [application settings](#).

Name	Name of the space or power source (not editable)
Value	The value of the boundary condition of specific space (temperature $^{\circ}\text{C}$) or power source (power density W/m^3).
Unit	Unit of the value of that specific boundary condition (not editable)
Rel. Humidity	Relative air humidity in percent
Part. Pressure	Partial pressure of vapour (in hPa) resulting from the temperature and relative humidity values entered (not editable)
Expose graphical evaluation too	If checked here, the graphical evaluation (Results 3D window) will be started after boundary conditions are applied. If this setting is turned off the graphical evaluation (if not open yet) will not be executed automatically. See also Application settings .
Evaluate secondary functions	If checked here, secondary functions (like heat flux, relative surface humidity, etc.) will be calculated for graphical evaluation. If turned off (unchecked) only temperature field will be calculated for Results 3D evaluations. This saves significantly time needed for calculation and reduces memory demand during evaluation. See also Application settings
Apply	Confirms data entered in this window end initiates the superposition followed by various evaluations.

You confirm the data entered by pressing the button *Apply*. Boundary conditions will be applied onto basic solutions (g-values). If calculation of basic solutions is also required it will be automatically initiated also.

MULTICORE-option: Speeding up computationally intensive jobs by distributing them on multiple processors or processor cores for parallel execution is only possible with an active MULTICORE-Option of the program.



Remark: The button "Apply" will blink if there is new solution set available from the [solver](#), but the data entered in this window has not been applied yet to the new solution set.

Remark: The button "Apply" is shown disabled if one or more basic solutions have not been calculated (solved) yet, i.e. they are missing.

The temperature distribution with the construction results from the superposition of respective basic solutions each multiplied by its boundary condition. Results are presented as [text reports](#) ready for printing or to be saved as Adobe PDF, MS.Word or MS-Excel files.

The distribution of temperature and further secondary functions is further used in various graphical [evaluations](#).

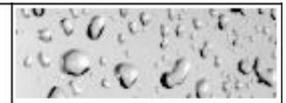
Templates of Boundary conditions are saved to [the project file](#) when the project is saved.

Remark: All BCs will be merged by their Names into the projects BC-Template upon apply. This will retain BCs even when spaces or power source are renamed or removed from the project. By that the BCs of other projects can be merged and saved - even such with names which are not yet (or no more) available if the current project.

Important: To retain the values of boundary conditions between program executions [the project must be explicitly saved](#) (project data is not saved automatically when boundary conditions are applied!).

Remark: You will need the value of power density W/m^3 to properly enter the boundary condition specific to power sources. For verification purposes you will find the total volume of each power source within [Modelling Report](#). Within [Results Report](#), in addition to the value of respective power densities, you will also find the overall power of each power source.

VAPOUR-option: Analysis of multidimensional vapour diffusion is only possible with an active VAPOUR-Option of the program.



Transient Boundary Conditions (periodic, harmonic)

For the purpose of dynamic, transient problem (subject to the TRANSIENT-Option) the boundary conditions window mutates to input of time dependant quantities.

Boundary Conditions - Period 86400 s.

Time Point to evaluate: 50400 s (14:00:00) Point Timelines Animate

Name	#H	#Hc	Periodic Data	#Hbc	#Hsynth	#Heff	Value	Rel.Hum	Part.Pres
aussen.de	1	1	ConstantValue	0	All ...	0	-10 °C	80 %	2,075 hPa
innen	1	1	RegularPoints	12	All ...	1	-48217,5156 °C	53 %	11554... hPa

Expose graphical evaluation too
 Evaluate secondary functions
 Display Surface Temperature (only for 2D projects)

Apply Boundary Conditions

TRANSIENT-option: Solving and evaluating time dependant dynamic, periodic problems when heat capacity effects are concerned is only possible with an active TRANSIENT-Option of the program.



The transient boundary conditions are provided as sets of harmonic Fourier coefficients created for respective periodic data (these are managed in [Periodic / Harmonic Data Editor](#)) for each and any Main Period of the transient problem (as selected in [Solver parameter](#) form).

Boundary conditions are kept distinct of each solution period and also saved to [the project file](#) when the project is saved.

Important: To retain the values of boundary conditions lately applied the [project must be explicitly saved](#) (project data is not saved automatically when boundary conditions are applied!).

Remark: All BCs will be merged for each Period (including MainPeriod=0 - steady state) by their Names into the projects BC-Template upon apply. This will retain BCs even when spaces or power source are renamed or removed from the project. By that the BCs of other projects can be merged and saved - even such with periods and names which are not yet (or no more) available if the current project.

Main Period (title-bar)	The Boundary Conditions window title bar displays the Main Period of the currently evaluation transient, periodic problem (as selected in Solver parameter form).
Time Point to evaluate	Some evaluation time point T within the Main Period of the solution. If checked the change to the time point is immediately applied together with boundary conditions and reflected in evaluation windows (Results report , Results 3D window) once these are open.
#H	Number of harmonics solved (harmonic solutions) for the selected Period and available to further evaluation
#Hc	Number of continuous harmonics solved for the selected Period and available to further evaluation
Periodic Data	Periodic boundary conditions data managed in Periodic / Harmonic Data Editor By default the Constant Value Periodic data will be initially set respectively to the steady state boundary condition.
#Hbc	Number of harmonics available from Fourier Analysis out of periodic boundary conditions data (managed in Periodic / Harmonic Data Editor) available to further synthesis and evaluation
#Hsynth	Number of harmonics chosen for use in harmonic Fourier Synthesis out of those available from periodic boundary conditions data (#Hbc) The highest number of harmonics used during synthesis (#Hsynth) results from the smallest count of harmonics within the base solution and out of the respective boundary condition and can be further reduced (-1 - all available). Remark: Evaluating in TRANSIENT mode with only 0-th harmonic or only Constant Boundary Conditions is identical to Steady State results (regardless of HARMONIC or TRANSIENT license feature).
#Heff	Effective number of harmonics used in harmonic Fourier Synthesis out of those available from periodic boundary conditions data (#Hbc) and those available from the solution set (#H).
Value	Synthetic boundary condition value at the evaluation time point T
Timelines	Opens Timelines window displaying synthetic temporal, time dependant results at selected Probe Points together with timelines of synthetic boundary conditions applied.
Animate	Displays the Animate Time Dialog used to define temporal animation of transient evaluation by automating changes to the evaluation time point T .

Time Point to Evaluate: provided existence of periodic boundary conditions on can set the evaluation time point (within the main period)

Remark: During graphical evaluation [the time point information](#) will be displayed in the [title of the colour bar](#).

For the evaluation the harmonic base solutions are combined with the respective harmonic coefficients of the boundary condition the belong to (complex linear combination). The resulting distribution harmonic coefficients it then passed for Fourier synthesis for specific [evaluation time point T](#).

Remark: Provided that the periodic boundary conditions did not change, only synthesis for [new time point](#) must be executed.

Remark: Evaluating in TRANSIENT mode with only 0-th harmonic or only Constant Boundary Conditions is identical to Steady State results (regardless of HARMONIC or TRANSIENT license feature).

Remark: The input of relative air humidity (when vapour diffusion problem has been solved too) are still constant (steady state) and independent of air temperatures. Only the partial pressure will be recalculated for respective, [evaluation time point](#) dependant, air temperature.

See also: [Element Editor](#), [Element types](#), [Evaluations and Results](#), [Standard Boundary Conditions](#), [Periodic / Harmonic Data Editor](#), [Animate Time Dialog](#), [Timelines Window](#)

Periodic / Harmonic Data Editor

The Periodic / Harmonic Data Editor window provides the means of creating, editing and managing periodic climate data required as boundary conditions to transient evaluation.

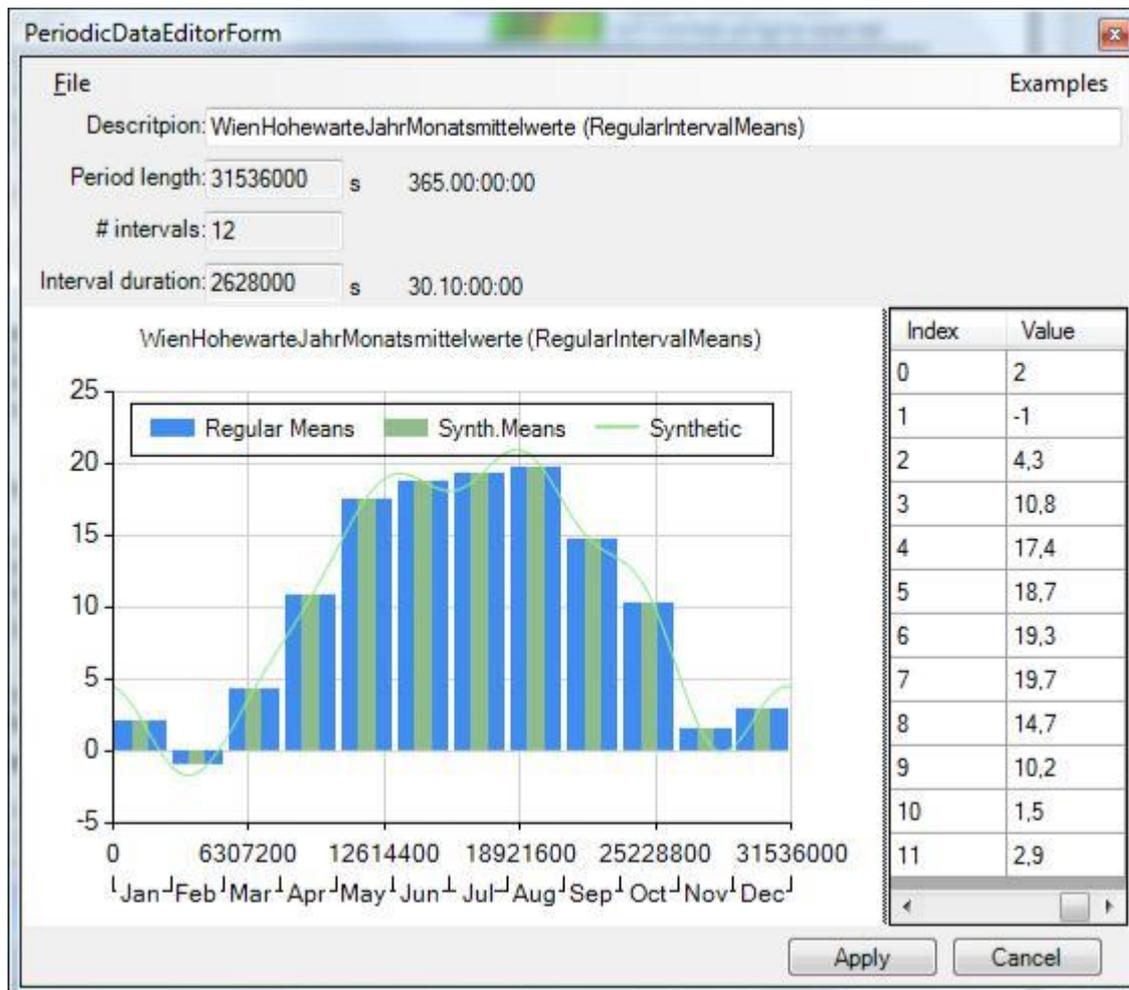
TRANSIENT-option: Solving and evaluating time dependant dynamic, periodic problems when heat capacity effects are concerned is only possible with an active TRANSIENT-Option of the program.



The Periodic / Harmonic Data Editor will be used out of the [Boundary Conditions window](#) when transient problem is to be evaluated and time dependant boundary conditions are needed for that purpose. For convenient managing periodic data sets it is available from a [Tools Menu](#) also.

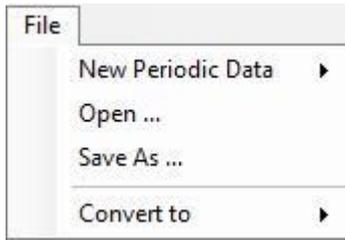
There are following types of data which could be managed within Periodic / Harmonic Data Editor:

- Constant
- Regular Points
- Regular Means
- Irregular Means
- Irregular Steps
- Harmonic Coefficients.



The window contains three parts:

- File & Examples Menu
- Info
- Graph
- Data Grid



File Menu:

- New - create new data set of the specific part. A dialog is shown to provide the information required for specific type (see "Info Part").
- Open/Save/Save As... - provides the file access to the data
- Convert To... - allows simple synthetic conversions between selected periodic data types.

Info Part:

The Info part displays the characteristic information for the dataset being processed. Depending on the type of data different information is displayed and eventually available to input.

The initial specific values are supplied during the creation of the data set (File->New...) and a limited set of them, depending on the data type processed, can be changed later.

- Description identifying the data
- Period Length
- Number of Intervals

Graph Chart Part:

The graph chart represents and displays the input data along with its synthetic variant resulting from harmonic Fourier analysis and synthesis.

Zoom with lasso...

The legend provides the identification of patterns shown. A double click on to the legend will change its position relative to the chart area (right, left, top, bottom).

Data points can be selected.

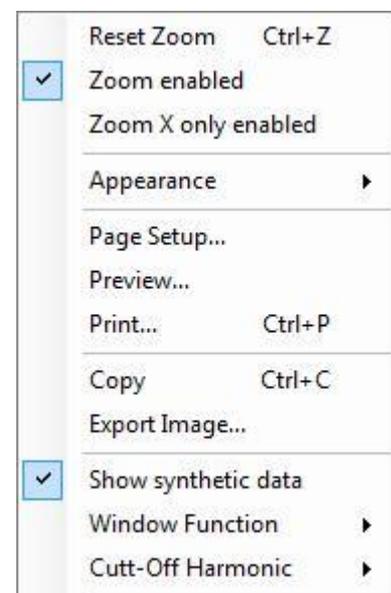
Data points (or bars) can be dragged changing the value (press ESC to abort the unintentional drag operation and revert to the previous value).

MULTICORE-option: Speeding up computationally intensive jobs by distributing them on multiple processors or processor cores for parallel execution is only possible with an active MULTICORE-Option of the program.



Context menu of the Graph Chart:

- Reset Zoom
- Zoom enabled
- Zoom X only enabled
- Appearance
- Page Setup...
- Preview...
- Print...
- Copy
- Export Image...
- Show synthetic data
- Window Function
- Cut-Off Harmonic



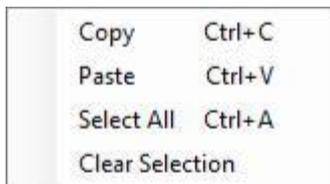
Window Function (Gibbs Effect)	<p>Selection of the windowing filter function to reduce the Gibbs effect (e.g. while working with switch functions "IrregularIntervalSteps" or dense data with high value changes).</p> <ul style="list-style-type: none"> • Fejer window • Hann window • Hamming window • None
Cut-Off Harmonic	Limits the number of harmonics for the Fourier synthesis.

Data Grid Part:

The data is input into the Value column.

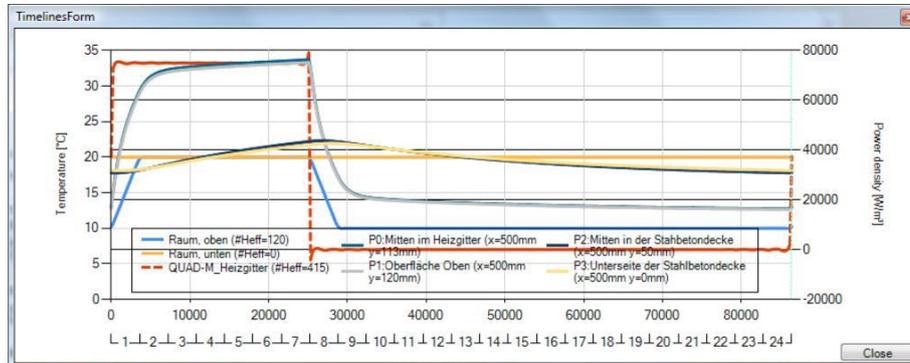
The native Copy & Paste of the selected cell(s) is provided.

Capability to paste data from/to the Value column streamlines the input procedure (also available via Context Menu). All numbers contained in the paste buffer (regardless if column(s) or row(s)) will be parsed as double entities (entries in error and not parse able as such in be ignored) and transferred in the order to the value column.



Timelines

The Timelines window displays synthetic temporal, time dependant results at selected [Probe Points](#) together with timelines of synthetic boundary conditions applied. The window is opened from the [Boundary Conditions window](#).



TRANSIENT-option: Solving and evaluating time dependant dynamic, periodic problems when heat capacity effects are concerned is only possible with an active TRANSIENT-Option of the program.



The creation of resulting chart involves of following work items which, depending on the complexity of the model, might be time consuming (during the calculation a progress bar shows at the bottom edge of the window):

- Timelines of boundary conditions are created by harmonic Fourier synthesis based on [effective number of harmonics](#) specified in the [Boundary Conditions window](#). For power sources a second vertical axis on the right side of the chart is displayed; this axis and curves showing the boundary conditions of power sources are display as dashed lines,
- Timelines of resulting values (temperatures) at selected [Probe Points](#) are generated by first computing synthetic distribution of results at specific time point for the whole model and then extracting the values at [specified by spatial](#) information.

Remark: Values are (re)computed for each time value corresponding to respective pixel or pixel pair position along the time axis. When the window is resized by increasing its width significantly (more then doubled) then the calculation is restarted.

MULTICORE-option: Speeding up computationally intensive jobs by distributing them on multiple processors or processor cores for parallel execution is only possible with an active MULTICORE-Option of the program.



The legend provides the identification of patterns shown. A double click on to the legend will change its position relative to the chart area (right, left, top, bottom).

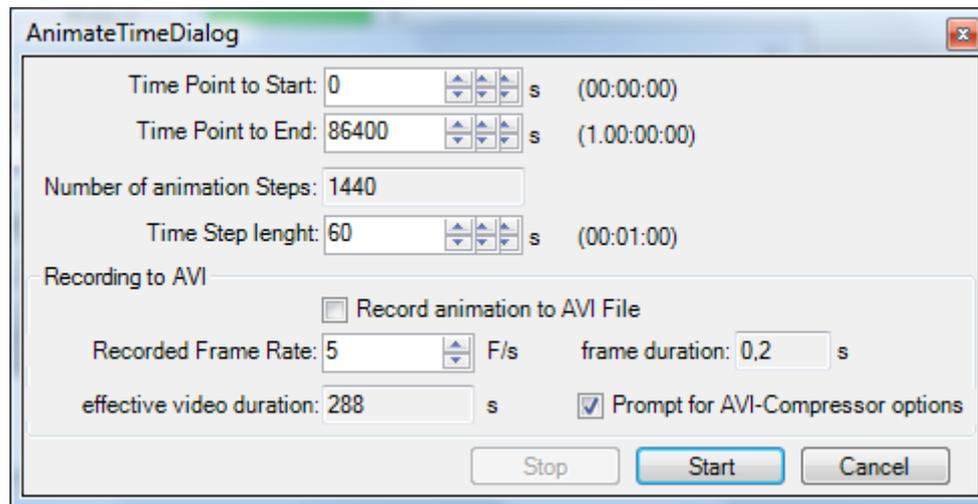
Animate Time Dialog

The Animate Time Dialog window is used to define temporal animation of transient evaluation by automating changes to the [evaluation time point T](#) of [Boundary Conditions window](#).

TRANSIENT-option: Solving and evaluating time dependant dynamic, periodic problems when heat capacity effects are concerned is only possible with an active TRANSIENT-Option of the program.



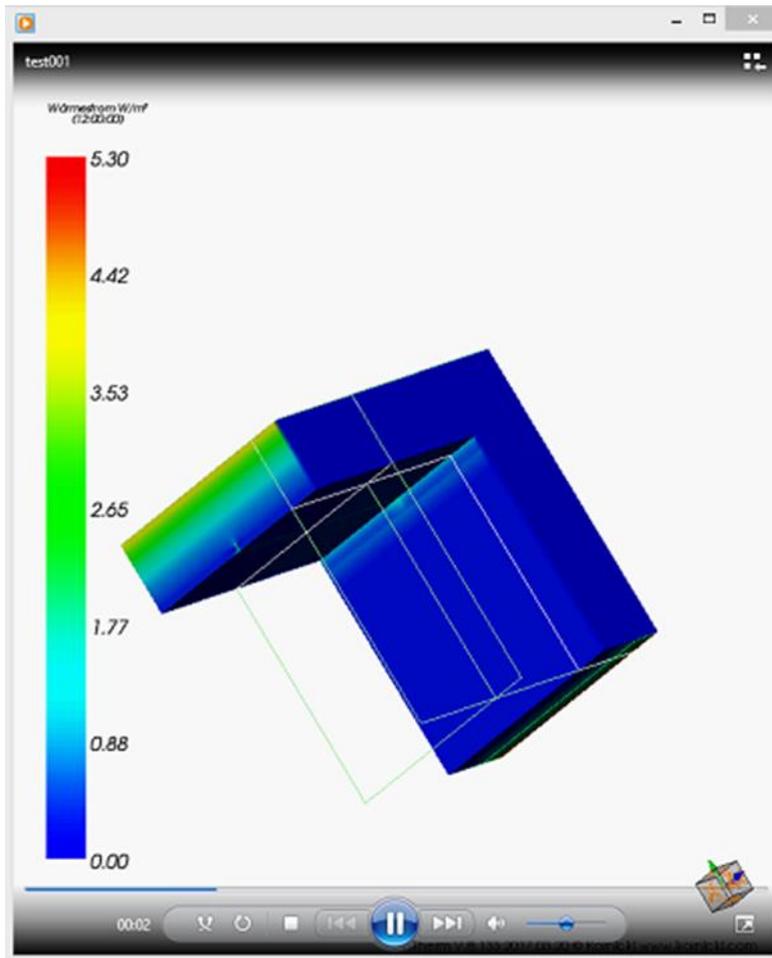
The temporal animation will be executed in specified time intervals continuously updating evaluation windows ([Results report](#), [Results 3D window](#)) once these are open an boundary condition applied. [Results 3D window](#) rendering can be optionally recorded to an AVI file.



Time Point to Start	The start value of iterating the evaluation time point
Time Point to End	The end value of iterating the evaluation time point
Time Step length	The step to increment evaluation time point on each iteration
Number of animation steps	Number of animation frames resulting from above inputs

Remark: The AnimateTimeSettings for each MainPeriod of the latest animation started are retained within application settings (Start, End, Step, currently set in AnimationTimeDialog)

The Recording to AVI part of the window will be active if there is an active [Results 3D window](#) rendering only, otherwise it is shown disabled.



Record animation to AVI File	If checked the image frame created for each time step will be recorded to a file. Prior to start of recording a dialog will ask you to specify the file name to record to. If the checkbox is cleared the animations started is reflected visually in the results shown only.
Recorded frame rate	Specify the frame rate for later playback.
frame duration	The reciprocal value of frame rate. Each time step frame will get the specified duration (for later playback)
effective video duration	Overall duration of later playback of the created file (number of frames * frame duration).
Prompt for Compressor options	If checked the operating system provided compressor options will queried prior the recording to a file will be started.

Remark: This VideoRecordingSettings (FrameRate, PromptCompressorOptions; currently set in AnimationTimeDialog) are saved within application settings (of the latest recording started successfully).

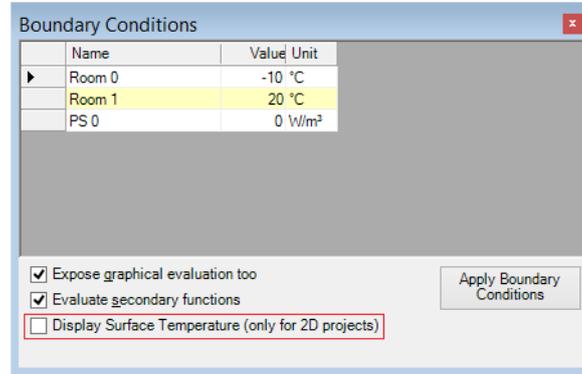
Start	Will start iterating the evaluation time point within the specified interval
Stop	Will stop the iteration.
Cancel	Will close the window (and stop the iteration too)

See also: [Results 3D window](#), [Results report](#), [Boundary Conditions window](#), [Timelines Window](#)

Surface Temperatures Window

In "View -> Evaluation & Reports -> Surface Temperatures" you can now, after applying the boundary conditions, access a graphical representation of the temperature gradients at the surfaces of the rooms the project contains.

You can also access this view by marking the option in the window "Boundary Conditions" and clicking "Apply Boundary Conditions".

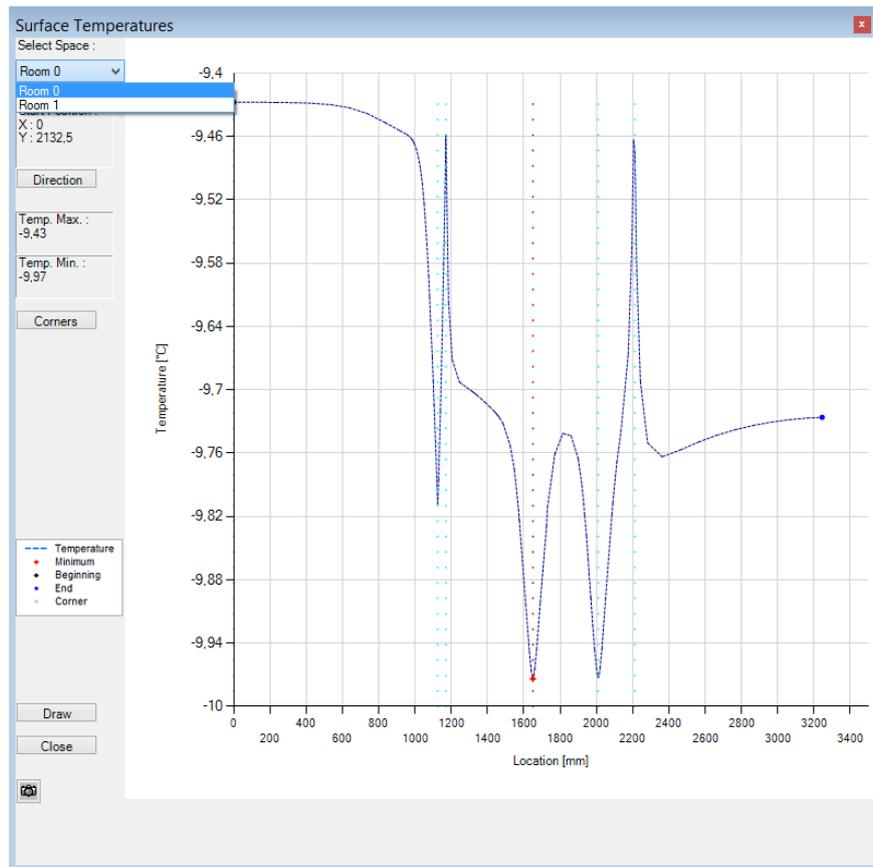


In the top left corner of the window "Surface Temperatures" you can choose the room for which you want to display the surface temperature gradient.

Click "Draw" to activate the graphical representation of the surface temperature gradient.

With "Direction" you can change the direction of the unrolling.

With "Corners" you can toggle the displaying of the corners.



Click the camera icon to create a Word document which you can process using Microsoft Word. In the window that is now opened you can mark the images which you want to transfer to the report builder.

Note: If you created more than four images buttons for scrolling will be displayed. A click always scrolls forward by four images. Then you can select the images which should be printed out with the report in the report builder after clicking "Graphical Evaluation".

Results report

Ergebnis-Bericht

Kormicki, Dienstleistungen in EDV und IT
A-1230 Wien, Othellogasse 1/RH 82
Tel./Fax: +43-1-61517099
email: kormicki@ohelo.at

AnTherm
V.8.131 2016.03.18
(c) Kormicki, all rights reserved

09.06.2016

Beispiel zum zweidimensionalen ablaufenden Wärme- und Dampfttransport (Diffusion)
Ausschnittsmodell im Bereich der Flachfläche einer Hallenbadwand
Fugenwand zwischen Wohnraum und Hallenbad, einschichtige Backsteinwand mit 5 cm Kerndämmung
Berechnung des Wärme- und Dampfttransports
Februar 2006: K. Krec
Datei: T:\Benchmark\krec_test\Hallenbad_2D_anthem

Randbedingungen (Lufttemperaturen / Leistungen)

	Raumtemp. [°C]	Min.Temp. [°C]	Max.Temp. [°C]	Grenzfuchte [%]	f _{Rsi}
aussen	-12,00	-11,48	-10,45	100,00 %	
Hallenbad	30,00	26,49	29,06	81,54 %	0,92
Wohnraum	20,00	16,13	23,00	78,40 %	0,88

Verwendete Wärmeübergangswiderstände

Raum	Wert	Oberfläche
aussen	Rs=0,04 m ² /K/W	ext
Hallenbad	Rs=0,25 m ² /K/W	Hallenbad
Wohnraum	Rs=0,25 m ² /K/W	Wohnraum

Gewichte für den kältesten Oberflächenpunkt

	aussen	Hallenbad	Wohnraum
g(aussen)	0,887246	0,066585	0,121051
g(Hallenbad)	0,011401	0,861884	0,000079
g(Wohnraum)	0,001353	0,071531	0,878871

The *Results report* provides the output related to [boundary conditions](#) applied and shows results for coldest surface points of each respective space and includes their temperatures, coordinates and temperature weighting factors also.

Calculation of evaluation results dependant on boundary conditions is requested by applying these values from the [Boundary Conditions window](#).

The results offered by this report are used for the assessment of fulfilling (or failing) the [condensation](#) and [mould growth avoidance criteria](#) (see [EN ISO 13788](#)).

For each space the application automatically locates points of **lowest surface temperature** and outputs its coordinates. Also **g-values** (the weighting factors) are calculated and output.

Along temperatures of coldest surface points the application outputs respective **dew point values** (maximum non-condensing air humidity) also [\[1\]](#). Respective values are calculated according to formulas of partial pressure as defined in EN ISO 13788:2002.

The evaluation is executed on top of base solutions solved and superposed with respective boundary conditions (reported by the number of equations or cells solved) then further refined to the [super fine solution](#) (reported by the number of nodes evaluated).

Boundary conditions and critical results

For each space connected to the model following results information is offered:

- name of the space
- air temperature (the boundary condition of that space as entered within the [Boundary Conditions window](#) and applied)
- the lowest surface temperature θ_{Rsi}^* at the surface of that space (at the coldest surface point)
- the highest surface temperature at the surface of that space (at the hottest surface point)
- highest allowed relative air humidity with respect to the coldest surface point of that space to avoid surface condensation (condensing humidity, Condens. RH)
- the temperature factor f_{Rsi}^* with respect to the coldest surface point θ_{Rsi}^* (see remarks below).

	Airtemp. [°C]	min.temp. [°C]	max.temp. [°C]	Condens. RH [%]	f _{Rsi}
aussen	-12,00	-11,49	-10,32	100,00 %	
Hallenbad	30,00	26,61	29,09	82,13 %	0,92
Wohnraum	20,00	16,13	22,84	78,40 %	0,88

[The temperature factor f^{*}R_{si}](#) will be evaluated against the space with the lowest temperature θ_e (i.e. that space will be assumed to be "exterior") and calculated according to standard by the definition $f_{Rsi}^* = (\theta_{Rsi}^* - \theta_e) / (\theta_i - \theta_e)$.

Remark: In a two space case [temperature factors f^{*}R_{si}](#) will be output in addition to temperature weighting factors g - calculated for points of lowest temperatures of surface of each space. The temperature factor f_{Rsi}^* can be used as construction characteristic indicator only when two boundary conditions are applied. **When three or more boundary conditions (spaces, power sources) are involved then the standard requires using of temperature weighting factors g** because f_{Rsi}^* is not suitable in such cases!

Remark: The output of [temperature factors \$f^*_{Rsi}\$](#) will be suppressed if the [application setting "fRsi - Two-Space only evaluation"](#) is turned on and there are more than exactly two [temperature boundary conditions](#) involved in the model. If there are more than exactly two [temperature boundary conditions](#) and the [application setting "fRsi - Two-Space only evaluation"](#) is kept turned off, then the appropriate warning message is displayed within the report: "*Warning: fRsi evaluation with more than two temperature boundary conditions!*".

If heat sources are also modelled:

- the overall power of the heat source (calculated from the boundary condition entered and its volume)
- the power density (the boundary condition of that heat source as entered within the [Boundary Conditions window](#) and applied)
- the effective volume of that heat source

Remark: The assessment of further extreme values (e.g. temperatures) at all surfaces and within interiors of all power sources is provided within the [2nd part of this report](#) too.

Weighting Factors (g-values) of coldest space points

The weighting factors (g-values) will be output as a matrix for the coldest point of every space. Every column of the matrix displays g-values for the coldest point of the space surface named in the column heading.

	aussen	Hallenbad	Wohnraum
g(aussen)	0,987854	0,064001	0,121030
g(Hallenbad)	0,011647	0,865891	0,000090
g(Wohnraum)	0,000499	0,070108	0,878880

Weighting factors g are characteristic for respective points of the construction and their value is independent from any boundary conditions. The summation of g -values of a point multiplied by the respective boundary condition results in the temperature of that point. Under the assumption that the location of extreme poles will not change under different boundary conditions one can easily calculate the resulting temperature at these varied boundary conditions.

Remark: The sum over all g -values belonging to temperature boundary conditions at specific point (sum of values in one column without those belonging to heat sources) is always 1.

Remark: If the number of matrix columns output shall overrun the page width then the output of overrunning matrix columns is continued in groups one below the other.

Coordinates of coldest surface points

The listing of points of lowest space surface temperatures for each space shows point coordinates, the respective surface temperature, the dew point and the temperature factor f^*_{Rsi} also.

Space	x [mm]	y [mm]	z [mm]	Temp. [°C]	f^*_{Rsi}
aussen	947,8000	1245,0000		-11,49	
Hallenbad	548,5000	750,0000		26,61	0,92
Wohnraum	0,0000	2130,7000		16,13	0,88

f^*_{Rsi} - Mould growth- and Condensation assessment criterion is fulfilled.
Warning: f^*_{Rsi} evaluation with more than two temperature boundary conditions!

Condensation and Mould Growth Assessment Criteria

If requested by the application setting "[fRsi,min design factors - Warn if undercut](#)" (on by default) AnTherm will automatically evaluate the assessment of the f_{Rsi}^* values by comparing actual values of f_{Rsi}^* to their [design values for condensation free](#) and [mould-growth free](#) constructions (also set within [application settings](#)).

- shall the actual value of f_{Rsi}^* undercut the [design value for mould-growth free](#) construction, the indicator (*) is shown
- shall the actual value of f_{Rsi}^* undercut the [design value for condensation free](#) construction (and eventually mould-growth criteria too), the indicator (**) is shown.

In addition, if either of the design values are undercut at any space surface there will be warning message displayed at the end of this report:

- (*) Warning: $f_{Rsi}^* < XXXX$ - [Mould growth assessment criterion](#) not fulfilled.
- (**) Warning: $f_{Rsi}^* < XXXX$ - [Condensation assessment criterion](#) not fulfilled.

If the criteria is fulfilled on the other hand following confirmation will be printed:

- f_{Rsi}^* - [Mould growth-](#) and [Condensation assessment criteria](#) are fulfilled.

Remark: The output of [temperature factors f*Rsi](#) will be suppressed if the [application setting "fRsi - Two-Space only evaluation"](#) is turned on and there are more than exactly two [temperature boundary conditions](#) involved in the model. If there are more then exactly two [temperature boundary conditions](#) and the [application setting "fRsi - Two-Space only evaluation"](#) is kept turned off, then the appropriate warning message is displayed within the report: "*Warning: fRsi evaluation with more then two temperature boundary conditions!*".

Probe Points report

The *Probe Points report* is used for the output of temperatures at [chosen probe points](#) at currently applied [boundary conditions](#).

Results Report

Kornicki, Dienstleistungen in EDV und IT
A-1230 Wien, Othellogasse 1/RH 8/2
Tel./Fax: +43-1-6157099
email: tkornicki@cheil.at

AnTherm
V.8.130 2015.04.23
(c) Kornicki, all rights reserved

16.06.2015

TUTORIAL - 3-dimensional:
Corner construction over carpet. Masonry bearing wall on girder over carpet and bonded with concrete floor slab (= ext. insul., perimeter insul., elect. bearing assembly in floor topping).

File: T:\Benchmark\TUTOR3_orig.anthem

Number of evaluated cells: 136954 (Nodes > 1095632)

Coordinates (x,y,z) for coldest surface point of each space

	x [mm]	y [mm]	z [mm]	Temp [°C]	f _{Rsi}
Room 0	0,0000	35,0000	1175,0000	-10,00	
Room 1	345,0000	1310,0000	1180,0000	13,91	0,80

f_{Rsi} - Mould growth- and Condensation assessment criterion is fulfilled
Warning: f_{Rsi} evaluation with more than two temperature boundary conditions!

Boundary conditions (Air temperatures / Powers)

	Airtemp [°C]
Room 0	-10,00
Room 1	20,00

	Power [W]	Power density [W/m²]	Volume [m³]
PS 0	82,98	6000,00	0,01382976

Temperatures at chosen points

x [mm]	y [mm]	z [mm]	T [°C]	
0,0000	35,0000	1175,0000	-10,00	min. Room 0 -10°C
405,0000	895,0000	1175,0000	-6,55	max. Room 0 -10°C
345,0000	1310,0000	1180,0000	13,91	min. Room 1 20°C (f _{Rsi} =0,8 (67,98%))
1483,7500	1310,0000	0,0000	27,42	max. Room 1 20°C
349,0000	1250,0000	1176,0000	13,05	min. PS 0 6000W/m²
1483,7500	1252,5000	60,0000	29,43	max. PS 0 6000W/m²

AnTherm - Heat & Vapour Transfer-Thermal Bridges- Walter, Udo, Klaus, Tomasz
Copyright (c) 2003-2015 M.Kornicki Dienstleistungen in EDV & IT http://www.kornicki.eu

2

The report outputs temperatures at points (coordinates x,y,z) defined in the window ["Probe Points"](#).

For each point given the application outputs coordinates (values of x,y,z) and the value of calculated temperature. If required a tri-linear interpolation is used to calculate temperature exactly at any point of the model from node values available at the [super fine grid](#).

If the chosen point is located at some space surface the respective dew point (maximum non-condensing air humidity) is calculated and output in %.

If the chosen point is located outside of the boundaries of the construction only its coordinates are output.

Remark: The report includes points of lowest and highest surface temperature for each space surface automatically – space name, space temperature, dew point and f_{Rsi} value (if applicable) are output – same as in [Results report](#).

Remark: The output of [temperature factors f*Rsi](#) will be suppressed if the [application setting "fRsi - Two-Space only evaluation"](#) is turned on and there are more than exactly two [temperature boundary conditions](#) involved in the model.

Remark: If there are power sources contained within the model this report also includes points of lowest and highest temperature within each power source (to assess overheating for example).

If there is solution of vapour diffusion available also, values of saturation and partial pressure are also output at each point:

Beispiel einer Wärmebrücke, die den Außenraum (Raum 0) und drei Innenräume (1,2,3) verbindet.
(2-dim. Beispiel)
Hergestellt im Büro für Angewandte Mathematik, A-1010 Wien.
File: D:\Entw\Walter\WalterWorkDir\dampf_AMIR_K2.antherm
Number of evaluated cells: 3180 (Nodes = 39201)

Boundary conditions (Air temperatures and relative humidities / Powers)

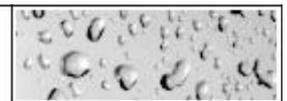
	Air temperature [°C]	Rel.Humid. [%]	Part.Press. [hPa]
Room 0	-12,00	80,00	1,7341
Room 1	20,00	53,00	12,3858
Room 2	20,00	53,00	12,3858
Room 3	20,00	53,00	12,3858

Temperatures and Vapour Pressures at chosen points

x	y	z	T [°C]	Sat.Press. [hPa]	Part.Press. [hPa]	
2158,0000	949,7300		15,32	17,40	12,31	3D collected
2158,0000	2163,5400		-2,98	4,76	7,77 *	3D collected
0,0000	2350,0000		-11,39	2,29	1,73	min. Room 0 -12°C/80%
2395,0000	3350,0000		17,24	19,67	12,39	min. Room 1 20°C/53% fRsi=0.91 (84.15%)
2320,0000	1505,0000		13,00	14,97	12,39	min. Room 2 20°C/53% fRsi=0.78 (64.05%)
1940,0000	1940,0000		16,52	18,79	12,39	min. Room 3 20°C/53% fRsi=0.89 (80.4%)

If the current value of partial pressure is higher than saturation pressure, a star (*) is shown adjacent to the partial pressure value - this shall alert on possible condensation risk at this respective point.

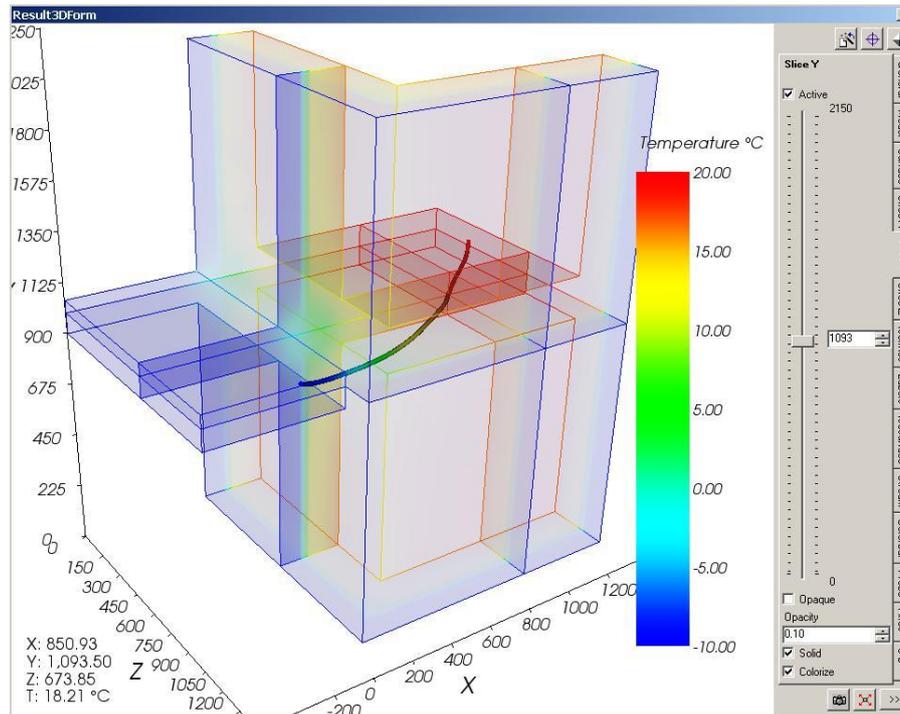
VAPOUR-option: Analysis of multidimensional vapour diffusion is only possible with an active VAPOUR-Option of the program.



Remark: Point evaluated are defined in the [window Probe Points](#). The list of probe points can be extended directly from [Results 3D window \(Probe\)](#) also.

Remark: Shall partial solution be only available to the report (e.g. some base solution yet not calculated finally, spaces without thermal connection with the model, etc.) the respective result fields will be left empty. Please consider checking the input data or rerunning the solver and checking the [output shown within the solver window](#) in such situation.

Results 3D window



The Results 3D window displays graphical 3D evaluations according to control parameters set in the [control panel](#) of this window. Changes to control parameters are immediately applied and result in new graphical image displayed. The 3D view can be arbitrarily manoeuvred [with the mouse in a 3D virtual space \(rotations, scaling or translations\)](#).



The image of the resulting 3D view can be rendered to the operating system's *clipboard* and then used in further applications (e.g. Word) to integrate it in personalised reports for example. It can be [saved to an image file](#) or [exported as a 3D scene file](#) too.

The window is divided into two areas:

- the 3D view
- the control panel
-

The 3D view shows the visualization of the 3D evaluation. Typically the actual view is overlaid with the current [colour bar](#) and with results of [probing evaluation](#) (shown at bottom left). The *orientation marker* (shown at bottom right) provides feedback of current orientation of axes within the 3D space. Additional output of program version (bottom right) and license information (top left) will display in the colour being contrast to current [background colour](#).



Actually the 3D evaluation displays a composition of possible [graphical evaluations](#) which are controlled selectively from the control panel. The arbitrary [composition of these parameters can be collected](#) for later reuse.

Remark: The position of the control panel can be set in [application settings to left instead of right side](#) of the window.

Remark: Within [application settings](#) you can decide if the [recently used combination of evaluation settings will be restored](#) on initial display of this window.

TRANSIENT-option: Solving and evaluating time dependant dynamic, periodic problems when heat capacity effects are concerned is only possible with an active TRANSIENT-Option of the program.



Evaluation of time dependant results of transient problem is controlled either by the [evaluation time point T](#) input in the [Boundary Conditions window](#) or automated with the [Animate Time Dialog](#).

The Control Panel

The control panel is used for selective parameterisation of any of 3D evaluations and for combining them in to visualizations within the 3D area. The graphical display resembles changes to various parameters immediately.

The final visualization depends on settings of all active evaluations, the opacity of their visual parts and choice of solid or wireframe views.

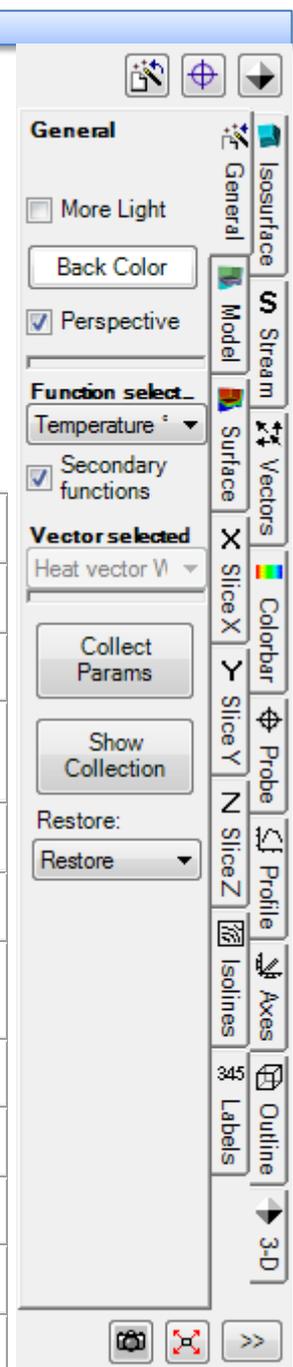
Which values if temperature, heat flux, humidity and others are evaluated depends on the choice of the active function in the General panel.

Names of respective control panels correspond to the evaluation controlled therein:

General 	General control parameters like selection of the active function evaluated , of light , of background colour .
Model 	Control parameters of the visualization of construction's model (similar to Elements 3D window)
Surface 	Controls parameters of evaluations specific to component's surface .
Slice X X Slice Y Y Slice Z Z	Control parameters 3D slices (parallel to axes planes) through the building construction
Isolines  (Isotherms)	Control parameters of isolines (isotherms, lines of equal heat stream density or surface humidity)
Streamlines S	Control parameters for evaluation of (heat-)streamlines within the construction over the vector field of (heat) stream density
Vectors  HedgeHog Arrows	Control parameters for evaluation of (heat-)stream vector arrows within the construction over the vector field of (heat) stream density
Isosurface 	Control parameters of a isosurface connecting points of equal values
Colorbar 	Selection of parameters controlling the colourizing of evaluations by the colour table chosen.
Axes 	Control parameters of the display of coordinate axes
Probe  Probe Cross	Control of the view of results of point probing . Particular probe can be selected from the list of probe points
Profile 	Control a X/Y diagram display of values along a profile line
Labels ³⁴⁵	Control properties and density of labels and annotations of isolines and the colour bar
Outline 	Control parameters of the display showing the volume's outline .
3-D 	3-D navigation within the three dimensional virtual view space (e.g. rotations, plan views etc.)

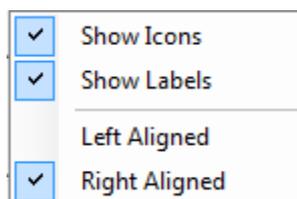
Buttons always accessible:

 Collect/Show Parameters	Complete combinations of evaluation parameters set and selected can be collected in the list of Results3D parameters and makes them available for later, single step, restoration. Remark: Clicking on this button while holding the <i>Shift-</i> or <i>Ctrl-Key</i> will simply expose the list of Results3D parameters without collecting current evaluation settings (adding the new entry).
 Collect/Show Probepoint(s)	Current coordinates of the probe point (actual position of X, Y, Z slices) are collected to the list of probe points . Remark: Clicking on this button while holding the <i>Shift-</i> or <i>Ctrl-Key</i> will simply expose the list of probe points without collecting the current point (adding a new



	entry). See also: Probe (control panel)
 Collect/Show Isometry	Current isometry (the 3-D transformation) is collected to the liste of isometries . The list of isometries can be edited in the isometries window or a chosen isometry can be selected and applied to the view. Remark: Clicking on this button while holding the <i>Shift-</i> or <i>Ctrl-Key</i> will simply expose the list of isometries without collecting the current camera position (adding a new entry). See also: 3-D navigation (control panel)
 Fit	The position and scaling of the view will be adjusted to ensure that the whole component is visible in the window.
 Copy Image	The 3D view will be rendered to the clipboard of the operating system . The image can be <i>pasted from the clipboard</i> to another application (e.g. Word) for further processing (e.g. integration in a personalized report). The image can be exported (saved) into an image file . Remark: The size of the image rendered is different from the view on screen and is defined within application settings .
 Record Video	What happens inside the Results 3D window is recorded and saved as a video file in the AVI format. To stop recording click this button another time.
[>>>], [<<<]	Hides- and exposes the control panel itself

The width of the control panel can be adjusted by dragging its border. Place the mouse pointer over the splitter edge of the control panel (left edge) until the mouse pointer changes to  . Then hold the mouse button and drag the edge of the control panel adjusting its width upon to your needs.
Remark: The chosen width will be saved when the window is closed and restored on the next display provided the the value is within the allowed range (116 to 600 display points).



Placing of the tabs of the control panel and their labels can be set to *right* or *left*. Adjust the setting "[Left-align vertical tabcontrol tabs](#)" within [application settings](#) accordingly.

The tabs can be shown [labelled with text titles](#) and/or displayed [with icon symbols](#) (controlled with context menu of vertical tabs). The setting "with Icons" only ("text labels" disabled) provides a very compact display of the tabs with only graphical symbols. All tabs show their respective tooltip also (which might contain further information). By that, even without text labels, the navigation in the control panels of Results3D is comfortably possible (and does not require any searching in several columns possible)

The control panel can be hidden or exposed by clicking onto the button [>>>] or [<<<] respectively.

Following control settings are common:

- [Active](#)
- [Opaque and Opacity](#)
- [Colorize](#)
- [Fixed Range](#)
- [Solid or wireframe model](#)
- [Background colour](#)

Remark: Actual creation of 3D evaluations result in relatively high memory demands. [Systems](#) equipped with less then 512MB RAM will experience significant delays in execution due to extensive paging. The number of [secondary functions chosen for evaluation](#) significantly controls these demands.

Important: In the case memory demands cannot be satisfied by the system error message will be displayed. If the application is still responsive after such an error message please save your data immediately and restart the application.

Remark: Current position of the 3D view (the isometry given by the current view vector) will be retained when the window is closed and [saved when the application exits](#). The next time the window is opened the view [isometry will be restored if such](#)

[behaviour is set within application settings.](#)

Remark: During the time the system is busy with rendering the new evaluation view (possibly very complex calculations) the mouse pointer will change to clock symbol (when placed over the control panel).

Remark: When this window is focused following additional, window specific, functions are available from the [main menu](#):

- [View→3D navigation functions](#) (plan views, rotations, perspective etc.)
- [Edit→Copy Image 3D](#) to render the [image onto the clipboard](#)
- [File→Export→Image 3D...](#) to save the rendered [image to a image file](#)
- [File→Export→Scene 3D...](#) to save the rendered [3D scene into a 3D scene file.](#)

Remark: The [size and proportions](#) of the [image rendered to the clipboard](#) can be set within [application settings](#).

Saving and Restoring parameter combinations

Complete combinations of evaluation parameters set and selected can be collected in the [list of Results3D parameters](#) and makes them available for later, single step, [restoration with a double click](#). This provides the valuable ability to quickly restore parameter combinations earlier saved, thus allowing reproducible visualizations. **Nearly all** parameters set within control panels are retained there as list entries. [Parameters specific to evaluation function](#) are handled otherwise



Function specific evaluation parameters

Evaluation settings which are specific to the evaluated function (i.e. to different [active function, for example the temperature, heat flux, pressure,...](#)) will be collected internally for each [chosen evaluation function](#). When the selection of the active function is changed the settings specific the the one chosen will be restored respectively - for optimal and comparable visualization of results for different projects.

By that it can be ensured that settings dependant on values of specific function (e.g. colours of the colour table, applicable assignment of colours to value domain, step of isolines, labels' parameter, etc.) will be kept invariant, as far as possible, for all projects.

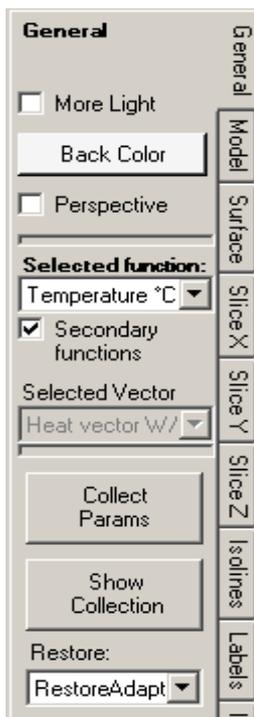
Active Function specific evaluation parameters:

Color Bar	Fixed Range (TotalRangeEditable), Range-Min/Max (TotalMinimum/MaximumValue), Min/Max (RangeMinimum/MaximumValue), Clamp range (Clamp), Colour table (ColorTable) and Label parameter (see below)
Isolines	Fixed Range (TotalRangeEditable), Range-Min/Max (TotalMinimum/MaximumValue), Min/Max (RangeMinimum/MaximumValue), Clamp range (ClampRange), #interv. (NoOfIntervals/Enabled), Step (Step/Enabled), Start At (StartAt/Enabled), Bold n-th (HighlightEachNth/Enabled) and Label parameter (see below)
Labels	Color Bar: # Intervals (ColorBarLabelsIntervals), Color Bar: Decimal digits (ColorBarLabelsDigits), Isolines: Font size [pts] (IsolinesLabelsFontSize), Isolines: Decimal digits (IsolinesLabelsDigits)
Isosurface	Value (IsosurfaceValue)

This function specific parameters, when set in the control panels, will be collected continuously for each evaluation function selected and saved within application settings when Result3D window closes.

Important: The function specific parameters do not run into the [collection of "Results3D Parameter"](#) (neither Project nor Standard). The function specific evaluation parameter are application setting.

General (control panel)



The control panel *General* is used to set and adjust general parameters of the visualization.

More Light	Used to turn addition "light sources" on making the the resulting view brighter and displaying more contrast.
Back Color	Defines the colour of the background used within graphical window
Perspective	Switches the 3D view between <i>perspective projection</i> and <i>parallel projection</i> . The parallel projection is especially useful during visualization of 3D models or slice views (see also Plan Views).
Function selected	By <i>selecting an active function</i> you define which data will be visualized. When additional secondary functions are calculated also (this is set with the switch „ Secondary functions “) one of following <i>functions</i> can be selected for the 3D visualization: <ul style="list-style-type: none"> • Temperature °C • Heat stream density W/m² • Relative condensing surface humidity % (only at the surface of the construction) according to EN ISO 13788:2002, respectively: <ul style="list-style-type: none"> • Condensation • Mould growth • Corrosion • Saturation pressure of vapour hPa • Partial pressure of vapour hPa • Vapour pressure difference hPa (saturation pressure - partial pressure) • Relative Core Humidity % (relative interstitial humidity; interstitial vapour saturation ratio = partial pressure/saturation pressure * 100) • Vapour diffusion stream density mg/m²h Which of these secondary function is actually calculated can be selectively defined within application settings by a parameter "Evaluate which secondary functions?" . If no secondary functions (beyond temperature) are available the field is shown disabled. 
Secondary functions	When the setting "secondary functions" is tuned off, only temperature distribution within the model is calculated and offered for evaluation. Turning this setting on requests calculation of additional scalar and vector functions. Which of these secondary function is actually calculated can be selectively defined within application settings by a parameter "Evaluate which secondary functions?" . Within application settings: Evaluate secondary functions
Vector selected	By <i>selecting an active vector field</i> you define which data will be visualized (e.g. during generation of streamlines or vectors (hedgehog, arrows)). When additional secondary functions are calculated also (this is set with the switch „ Secondary functions “) one of following <i>vector fields</i> can be selected for the 3D visualization: <ul style="list-style-type: none"> • Vector field of heat stream W/m² • Vector field of the vapour diffusion stream density mg/m²h Which of these secondary vector field is actually calculated can be selectively defined within application settings by a parameter "Evaluate which secondary functions?" . If there are no vector fields available the field is shown disabled. 
Collect Params	The current combination of numerous parameters set in all control panels will be saved to the list of Results3D parameters as new

	<p>entry.</p> <p>The list of parameters can be edited within Results3D parameters window (e.g. it can be named arbitrarily) or the required parameter combination can be selected from the list and applied to the current Results3D view. This provides the ability to quickly restore earlier saved combination of parameters allowing</p> <p>reproducible visualizations.</p> <p>The list of parameters is saved within project file.</p> <p>The Results3D Parameter window will be shown also.</p>
Show Collection	The Results3D Parameter window will be shown.
Restore	<p>Changes the application settings deciding if the recently used combination of evaluation settings will be restored on initial display of the Results 3D window.</p> <p>For more details see: Application setting "Restore last Result3D Parameters"</p>

The choice of [evaluation function selected](#) and [the vector selected](#) retained within application settings and restored respectively.

If the earlier selection is not available within the current data set (i.e. respective secondary function was not calculated) the selection will be reset to default. See also: [Selected Function specific evaluation parameters](#) and the [application setting "Restore last Result3D Parameters"](#).

Remark: As *primary function selected* we call the set of values used for current visualization. One can select the temperature, heat stream density, relative surface humidity etc. To reduce the time required for calculation, one can suppress the calculation of secondary functions with the [switch "Secondary functions"](#) (creation of data of each separate secondary function requires approximately same time as the creation of the temperature distribution alone when new boundary conditions are applied).

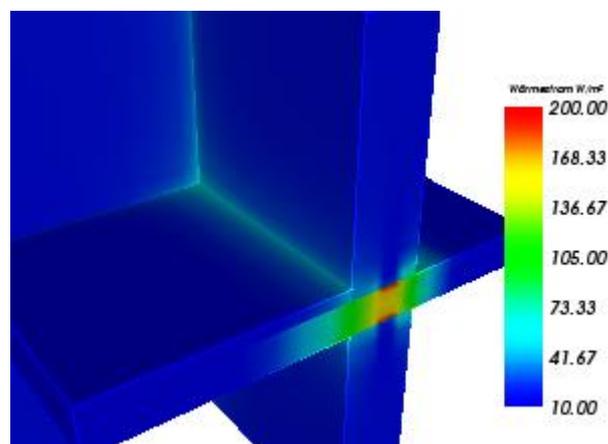
Calculating secondary functions can be significantly improved on multiprocessor systems ([MULTICORE-Option](#)).

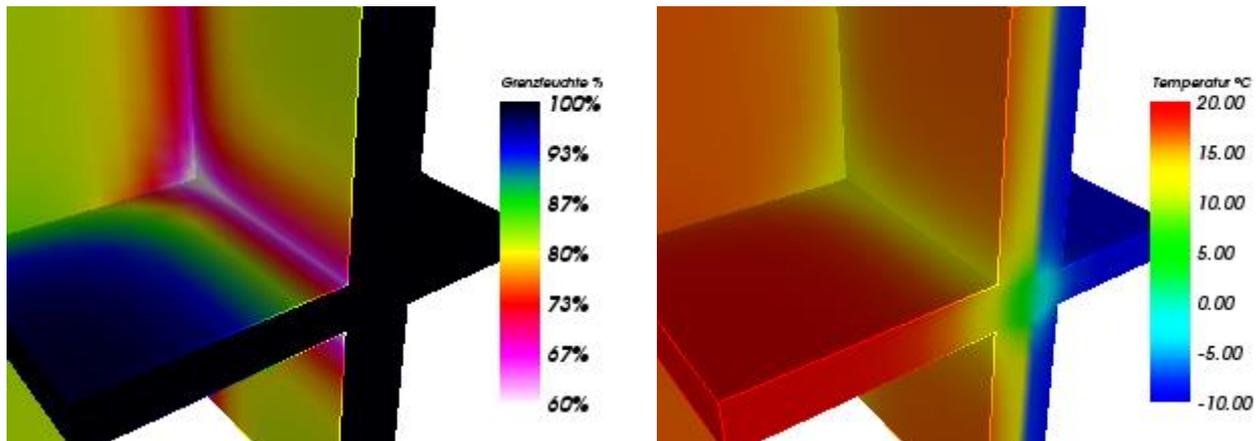
MULTICORE-option: *Speeding up computationally intensive jobs by distributing them on multiple processors or processor cores for parallel execution is only possible with an active MULTICORE-Option of the program.*



Remark: The relative surface humidity is defined only at the component's surface. At adiabatic boundaries (wonder foils) and at surface points hotter then the air space temperature the value of 100% is applied currently.

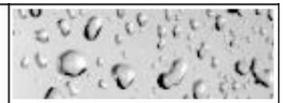
Following images display visualizations of different primary functions (Temperature °C , heat Stream Density W/m², Relative Surface Humidity %):





Remark: Producing a visualization of (heat) streamlines requires the calculation of the corresponding vector field of heat (or vapour) stream. Therefore you must turn the calculation of such required secondary vector field on to receive stream line visualizations (i.e. of [secondary functions](#)).

VAPOUR-option: Analysis of multidimensional vapour diffusion is only possible with an active VAPOUR-Option of the program.



See also: [Results3D window](#), [Active \(setting\)](#), [Opaque and Opacity \(setting\)](#), [Colorize \(setting\)](#), [More Light \(setting\)](#), [Background Colour \(setting\)](#), [Main menu](#), [Application settings](#)

Results3D Parameter window

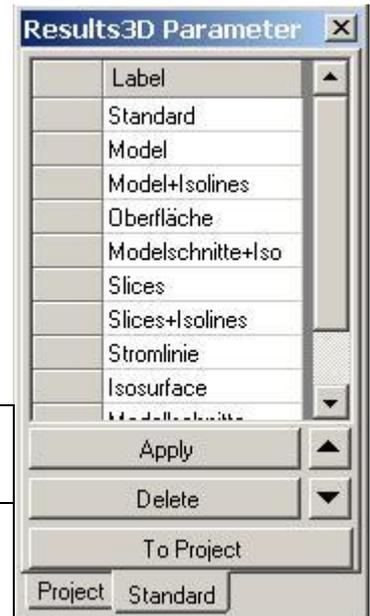
The window *Results3D Parameter* shows the list of collected 3D evaluation and visualization settings parameters. **Nearly all** parameters set within control panels are retained here as list entries. [Parameters specific to evaluation function](#) are not influenced from this window.

 Collecting new Results3D parameters is initiated from the [Results3D window](#) with the button [Collect Params \(in the control panel General\)](#) or from the [View menu](#).

The set of parameters shown as an item in the list can be easily restored and applied to the evaluation view.

The list of evaluation settings specific to the project is shown in the tab **Project**. The tab **Standard** shows the list of [application wide](#) evaluation settings.

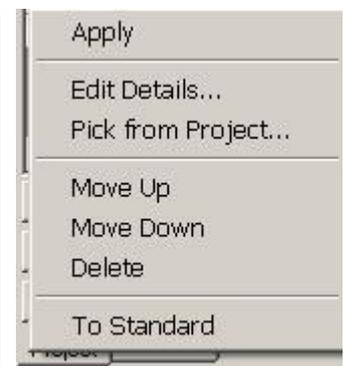
Label	An arbitrary, editable, label of respective item in the list of parameters (used for documentation and better, easier identification only)
Double-Click onto record-mark	Initiates the process of applying the combination of settings selected (marked line) to the 3D visualization. Remark.: Settings applied are taken from the current row – i.e. even if the click occurred within text of another line, the data applied is taken from the line marked with the record mark (an arrow symbol).
Apply	Initiates the process of applying the combination of settings selected (marked line) to the 3D visualization.
Delete	Deletes the currently selected (marked) item form the list.
Move Up ▲	The position of the item currently selected can be moved in the list.
Move Down ▼	The position of the item currently selected can be moved in the list.
To Standard	Settings combination associated with the selected entry within the project specific list will be copied (as additional entry) to the application wide list (Standard).
To Project	Settings combination associated with the selected entry within the application wide list (Standard) will be copied (as additional entry) to the project specific list.
Project	Exposes the list of project specific evaluation settings
Standard	Exposes the list of application wide (standard) evaluation settings



Functions of the context menu

The display of the list has a context menu attached (exposed when right mouse button is pressed):

Apply	Initiates the process of applying the combination of settings selected (marked line) to the 3D visualization.
Edit Details...	Setting combination of the currently selected item can be edited in detail and adjusted according to specific needs. Remark: This is an advanced function. Naming chosen for distinctive parameters shall be self explanatory.
Pick from Project...	The list of project specific evaluation settings saved within some another project will be imported (appended) to the current project's list.
Move Up	The position of the item currently selected can be moved in the list.
Move Down	The position of the item currently selected can be moved in the list.



Delete	Deletes the currently selected (marked) item form the list.
To Standard	Settings combination associated with the selected entry within the project specific list will be copied (as additional entry) to the application wide list (Standard).
To Project	Settings combination associated with the selected entry within the application wide (Standard) list will be copied (as additional entry) to the project own list.

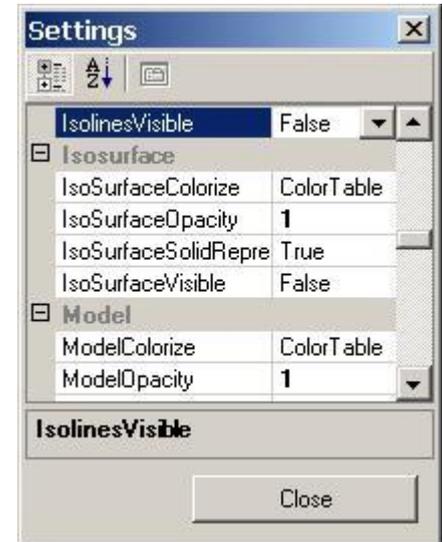
Please note: Various settings listed in this window can be set within other control elements also. The position of [slices X/Y/Z](#) can be set from [Probe points window](#) (see also [control panel Probe](#)).

The [camera position \(isometry\) of 3D view](#) can be set from [Isometries window](#) (see also [control panel 3D](#)).

Remark: To retain the list and the evaluation settings of the project between application executions you must [save the project!](#)

Remark: The application wide list of evaluation settings (Standard) will be [saved with application settings](#) (at application's exit).

See also: [Results3D window](#), [General \(control panel\)](#), [application settings file AnTherm.exe.Settings.xml](#), [Function specific evaluation parameters](#), [Settings - various \(dialog\)](#)



Model (control panel)

The control panel *Model* is used to adjust the parameters relevant for the visualisation of modelled building construction - in addition or in combination with other [evaluations](#).

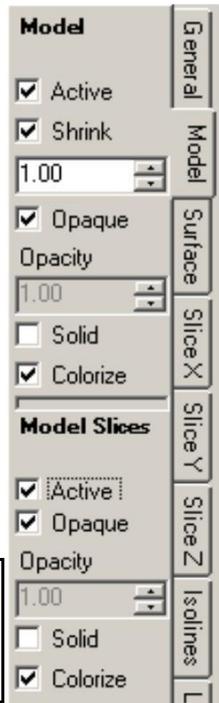
Following combination of evaluations might be of special interest:

- View of the model shown as [solid body](#) with [isolines](#) (e.g.. [isotherms](#)) on [component's surface](#),
- View of [slices through the model](#) (solid body) with [isolines](#) on slice planes,
- View of the model as lines (boundaries of materials) overlay-ed with other [evaluations](#).

The model representation shown corresponds to the one used in [Elements 3D window](#) within input part. The resulting image shows the modelled construction resulting from the overlapping of input elements (the entry point for the so called "minimum grid").

Surfaces of the model (and eventually edges) are shown colorized. Colorizing corresponds to the input of [element properties](#).

The [Slice planes X/Y/Z](#) can be also shown as model slices. Resulting representation can be shown as solid (colorized by material colour) or edge-lines (i.e. material edges).



Active , Opaque , Colorize	Decide about the visibility , colorizing and transparency of model surfaces (and its edges if applicable)
Shrink	Elements will be "shrunk" towards their centre (the value of 1.0 means no shrinking). Shrunk view is very helpful when examining the modelled construction in its interior also.
Solid	Allows the wireframe view of edges of the model input.

By default the construction model is shown as lines (wireframe) resulting from the input of elements.

The view onto edges of slice planes (these are located at the surface of the model) can be turned on or off within [the control panel „Surface“](#). This is important in the context of wireframe view of the modelled construction to either show lines coloured by material or by selected evaluation function.

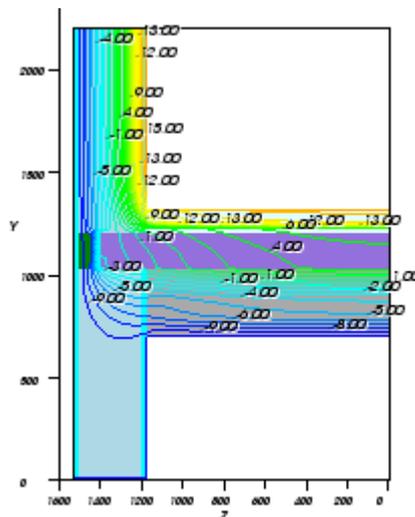
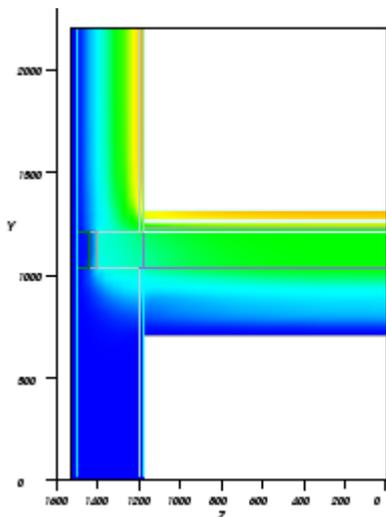
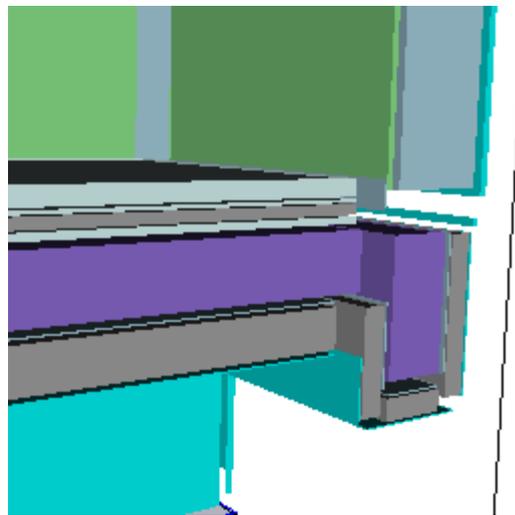
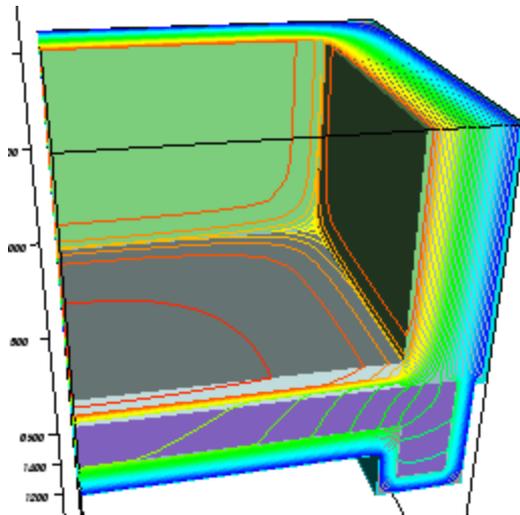
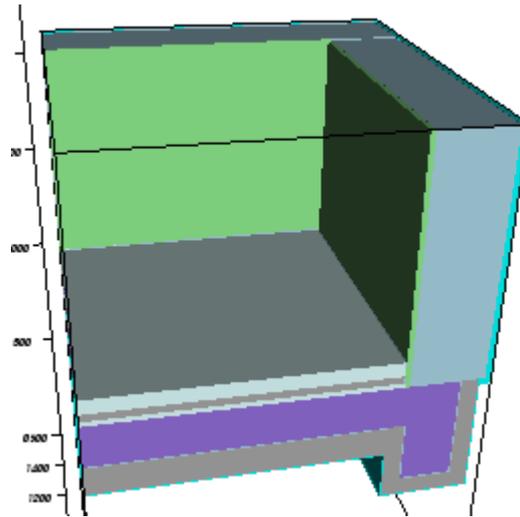
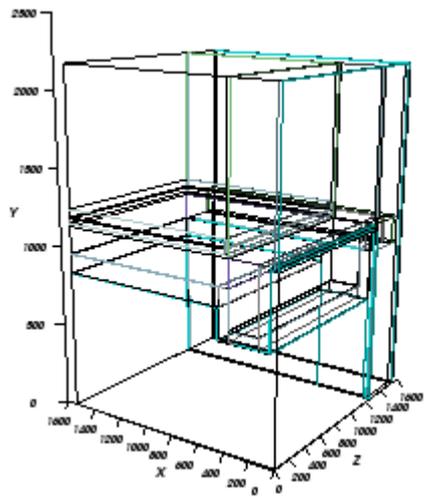
By default the view of model slices is turned off or set to wireframe. This is required to avoid overlapping of that representation with slices of selected function and to avoid confusing results and interpretation difficulties when too many lines are simultaneously shown during initial display.

Remark: Planes representing construction surface and model surface overlap. These can lead to visually "confusing" effect of "shadows" shown on the surfaces. Therefore it is advised to select wireframe representation for one of the two surface evaluations.

Important: Some (external) material edges overlap with surface edges and slice edges. If all these are turned on the overlapping might result in visually "confusing" effects.

Important: Model slice plane and function slice plane overlap. When both are turned on this overlapping will lead to sometimes confusing effect of "shadows". To make model slices visible at their own make slices itself fully transparent (i.e. turn their opacity = 0). If you wish to view slices through function values, it is advised to show model slices as lines (wireframe, solid off) to show edges of materials only.

Important: To properly show lines at material edges elements of the modelled construction are shrunk by a very small factor – otherwise the edges of neighbouring elements could not be shown properly (visually confusing result, possibly some lines even not visible). The shrinking factor used has a value of 0.99999, i.e. an element of 10 meters length is shrunk by less than 1 mm (only in the picture, which is absolutely invisible).



See also: [Results 3D window](#), [Active \(setting\)](#), [Opaque and Opacity \(setting\)](#), [Colorize \(setting\)](#), [Solid or Wireframe \(setting\)](#), [General \(control panel\)](#), [Isolines / Isotherms \(control panel\)](#), [Elements 3D window](#)

Surface (control panel)



The control panel *Surface* is used to adjust parameters of the surface visualization and thus has also impact onto results of evaluations related to the components surface: [Colorizing](#) and [Isolines \(Isotherms\)](#). The surface, if rendered opaque, will obscure all other evaluations specific to the objects interior ([Isosurface](#), [Streamlines](#), [Slices](#) etc.)

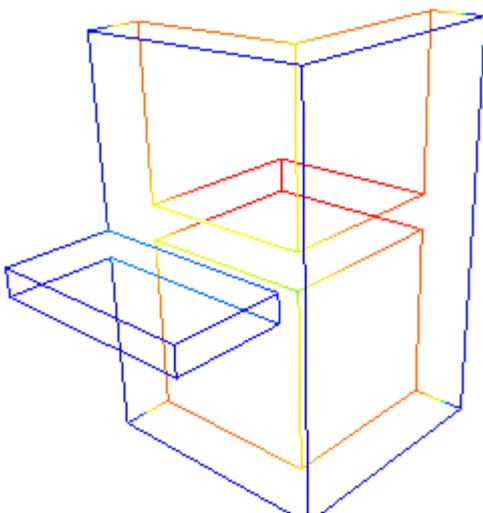
The surface (and eventually its edges) are rendered [colored](#). Actual colorizing follows values of currently [active function](#) (temperature, heat stream density, surface humidity etc. - selected within [General control panel](#)) and currently selected [Colour table](#).

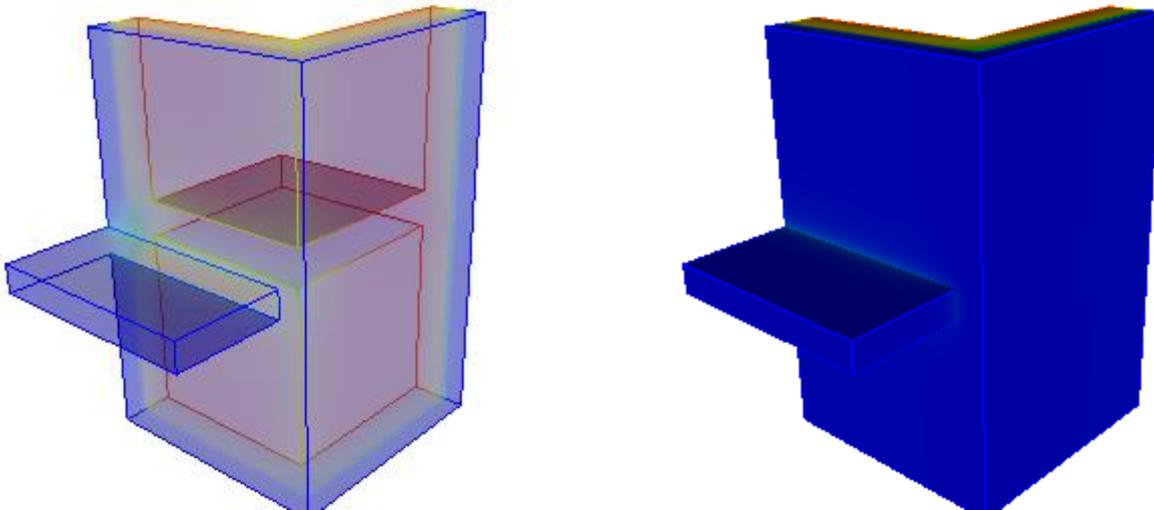
Active, Opaque, Colorize	Decide about the visibility, colorizing and transparency of the surface (and its edges if applicable)
Solid	When turned off allows the wireframe view onto the fine grid at which temperature distribution and secondary functions are calculated.
Edges	Allows emphasizing of component's edges, i.e. lines at intersections of surface plane elements. Rendering of surface edges can be requested independently of the surface - made visible or transparent .
Slice Edges	Allows emphasizing of slice edges (rendered only if any slices are active). Rendering of slice edges can be requested independently of the surface - made visible or transparent .

Remark: The surface displays components hull coloured by the [primary function](#) (including adiabatic cut-off planes – so called "wonders", adiabatic planes). The surface is rendered partly transparent (according to the setting of the [opacity value](#)). To receive an opaque (non transparent) rendering use the [switch Opaque](#). The surface can be rendered as a wire frame (lines) – turn the [switch Solid](#) off – to display the grid used for evaluation of results.

Remark: [Construction edges](#) show the position of all edges of the component. This rendering is useful while the surface itself is not shown (inactive or transparent). **Displaying construction edges might be also very helpful when verifying the correctness of model input.**

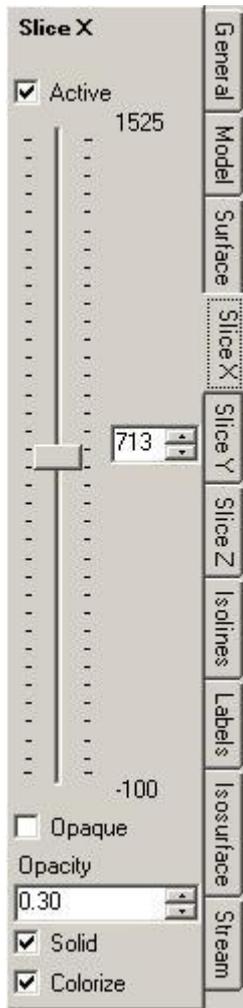
Remark: Graphical objects used to render the surface are oriented outside of components interior. This also ensures proper colouring of the wireframe view of the fine grid (the „back side“ of the wire frame is black).





See also: [Results 3D window](#), [Active \(setting\)](#), [Opaque and Opacity \(setting\)](#), [Colorize \(setting\)](#), [Solid or Wireframe \(setting\)](#), [General \(control panel\)](#), [Active function](#), [Isosurface \(control panel\)](#), [Streamlines \(control panel\)](#), [Vectors \(HedgeHog, Arrows\) \(control panel\)](#), [Isolines / Isotherms \(control panel\)](#)

Slice X, Y, Z (control panels)

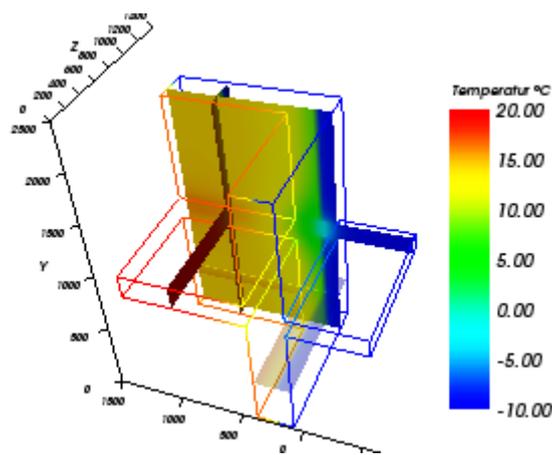


Control panels *Slice X*, *Slice Y*, *Slice Z* are used to adjust parameters of slice planes parallel to coordinate axes planes cutting through the construction. These settings have impact onto result of evaluations focused onto slice planes also: [Colorizing](#) and [Isolines \(Isotherms\)](#). The inputs are also used to control the [Probe evaluation](#), [Profile evaluation](#), [Streamlines evaluation](#) and [Vectors \(HedgeHog, Arrows\) evaluation](#).

Slice planes are shown [colorized](#). Actual colorizing follows the values of [active function](#) (temperature, heat flux (stream density), etc. - selected within [General control panel](#)) and currently selected [Colour table](#).

Crossing lines of the three slice planes (these lines are parallel to the axes) can be also shown (controlled via [Probe](#) panel). That other panel provides additional [control of interactive moving the cross point](#).

Active, Opaque, Colorize	Decide about the visibility , colorizing and transparency of the slice
Value	The slice position is defined by an input of plane coordinate (x for slice-X, y for slice-Y or z for slice-Z). The input can be performed either by <ul style="list-style-type: none"> • entry of numeric value in the input field or • moving the slider between the extent values of respective coordinate axis or • selecting a value from the context menu
Context menu/ Coordinate list	The context menu of this control panel offers the list of all surface coordinates within the modelled construction (in the direction of respective axis). By choosing one of the values the position of the slice plane will be set accordingly.



Remark: Because slice plane intersects with model's interior only it is advisable to either turn the view of the [surface](#) completely off or show it partially transparent (i.e. [not opaque](#)) or show it as [wireframe](#) only.

The tooltip of the tab will show up if the [surface](#) obscures objects rendered within the interior of the model.

Remark: The slice planes (SliceX, SliceY, SliceZ, orthogonal to respective coordinate axis) show plane intersections with construction colored by values of selected [function](#). If required values are interpolated (linear interpolation). Slices are shown partially transparent (according to current [opacity value](#)). To receive opaque representation (non transparent) use the [switch Opaque](#).

Remark: Slice edges (these are placed at construction's surface) are shown to emphasize the position of the slice plane even if the slice itself is transparent and not visible. The [exposure of slice edges](#) can be controlled from the [control panel Surface](#).

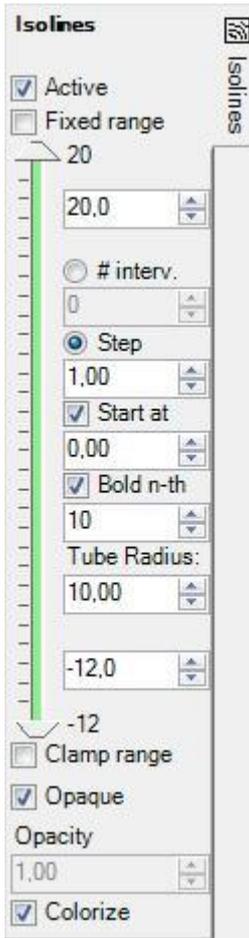
Remark: The position of SliceX/Y/Z will be retained within application settings and restored respectively.

If the old slice position is outside of the model bounds (with model spatially changed or different model loaded) the respective slice position will be reset to the centre of the model. See also [Application setting "Restore last Result3D Parameters"](#).

Remark: A double click onto the data record mark of a line corresponding to a probe point within the [probe points window](#) will initiate the transition of the position of the [probe point](#) currently shown within Results3D window to coordinates shown in the respective line – slice planes X/Y/Z are moved accordingly. Within a 2D project slice plane Z is moved to 500 mm.

See also: [Results 3D window](#), [Active \(setting\)](#), [Opaque and Opacity \(setting\)](#), [Colorize \(setting\)](#), [Solid or Wireframe \(setting\)](#), [General \(control panel\)](#), [Active Function](#), [Isolines \(Isotherms\)](#), [Probe \(control panel\)](#)

Isolines / Isotherms (control panel)



The control panel *Isolines* is used to adjust parameters of isoline evaluations (isotherms, in the case of temperature is chosen for the active [evaluation function](#)).

Isolines are shown colorized. Actual colorizing follows values of currently [active function](#) (temperature, heat flux, surface humidity etc. - selected within [General control panel](#)) and currently selected [Colour table](#).

The isolines (or isotherms) are drawn on [component's surface](#) or on [slice planes](#), and are therefore only visible if these evaluations are [active](#) also. Settings chosen within control panels [Slice X](#), [Slice Y](#), [Slice Z](#) and [Surface](#) have thereafter an impact onto evaluation of isolines.

Which isolines are shown can be selectively chosen by setting the number of intervals (#Interv.) or by choosing the step value. Isolines will be created only for values within the interval Min and Max according to the input therein.

At initial view of isotherms the interval of 1°C, starting at 0°C is set and isolines shown.

Isolines can be [labelled accordingly](#) also and/or [emphasized bolded \(as a tube\)](#).

Active, Opaque, Colorize	Decide about the visibility , colorizing and transparency of the lines shown
Fixed range	If turned on the value range for isolines' creation can be set arbitrarily, independent of the current value range of the evaluated function.
Range-Min/Max	The values of Maximum and Minimum, shown at the top and bottom of the panel, display the current extent of selected evaluation function. When " Fixed range " is chosen both values can be set arbitrarily, independent of the current value range of the evaluated function.
Min/Max	The interval can be reduced by the input in the respective fields. The input can be performed either by the entry of numeric values into the respective input field or by moving the sliders between the extent values.
#interv	Number of equidistant intervals dividing the extent between the input Min and Max values. The step will be calculated from these values.
Step	The value interval at which isolines shall be drawn.
Start at	The start value fro which (in both directions, towards Min and towards Max) the values of isolines to be drawn shall be calculated.
Bold n-th	Selected isolines (each N-th) will be shown emphasized (Bold n-th) - as a thick tube instead of a thin line. Isoline labels are shown only on bolded lines if the option "Bold n-th" is chosen. Remark: The tube itself if always partially transparent to allow proper display of isoline labels (visibility test)
Tube Radius	Emphasized isolines (switched on by " Bold n-th ") are shown as a tube to better emphasize respective isolines (compared to single pixel thick line). The radius is entered in units of the coordinate system - i.e. millimetres. Setting the tube radius = 0 will show it as simple line. The initial value of tube radius is computed based on median extent of component (1/200 of it). Remark: The number of sides rendered for the tube can be adjusted within application settings.
Clamp range	Limits the isolines created to the values within the Min/Max interval specified. If switched off (default) the isolines are created for the whole value range available in the data set.

Remark: It is advisable to adjust the surface "holding" the isolines (i.e. [Surface](#) or [Slices](#)) by showing it [transparent or opaque](#).

Alternatively adjust [colorizing](#) of isolines or the surface "holding" them to *dark* or *bright* to receive the required contrast of the image.

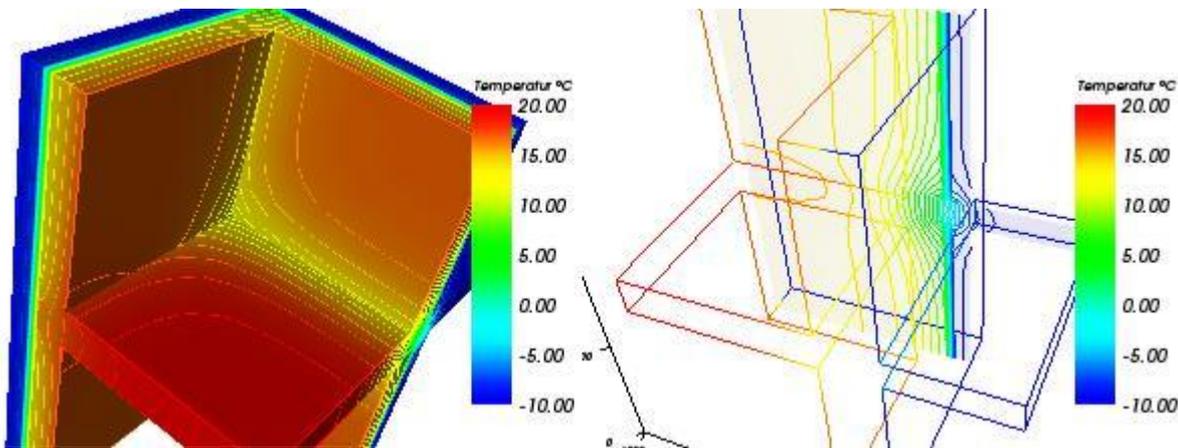
Remark: Because slice planes, holding isolines, intersect with model's interior only it is advisable to either turn the view of the [surface](#) completely off or show it partially transparent (i.e. [not opaque](#)) or show it as [wireframe](#) only. The tooltip of the tab will show up if the [surface](#) obscures objects rendered within the interior of the model.

Remark: Isolines are lines shown and colored by the constant value of the [primary function](#) chosen (Isotherms, lines of equal heat flux, lines of equal surface humidity, and so on). Isolines are displayed on [construction's surface \(Surface\)](#) and on [slice planes \(SliceX,Y,Z\)](#) (only if these are active). To show isolines only, without the surface "holding" them, turn that surface completely transparent (set [opacity = 0](#)). The selection of values at which isolines are shown is given by the definition of a start value (Start at) and of a interval length (Step) (or number of intervals).

Remark: The visibility of isolines is directly dependant on the visibility of the surface object holding it:

→ Isolines on the component's surface are only visible if the [Surface](#) is also active. To get better view onto isolines turn the surface transparent – Surface→Opacity=0.0;

→ Isolines on [slices \(X/Y/Z\)](#) are only shown if the respective slice is also active. To get better view onto isolines turn the respective slice transparent – SliceX/Y/Z→Opacity=0.0



Hint: [Fix the Range](#) to receive comparable results by decoupling it from the actually evaluated domain. This [function specific setting](#) takes care for keeping [the range constant](#) regardless of changes to [boundary conditions](#), [evaluation time point](#), [animation](#) or for different projects.

See also: [Results 3D window](#), [Active \(setting\)](#), [Opaque and Opacity \(setting\)](#), [Colorize \(setting\)](#), [Solid or Wireframe \(setting\)](#), [General \(control panel\)](#), [Active Function](#), [Isosurface \(control panel\)](#), [Labels \(control panel\)](#), [Fixed Range \(setting\)](#)

Labels ³⁴⁵ (control panel)

The control panel Labels contains settings for labelling of [Isolines](#) and of the [Colorbar](#)

Labels on Isolines

Labels of [Isolines](#) are placed over Isolines shown over the [slice planes X/Y/Z](#) or over [component's surface](#).

Show labels	Turns labelling of isolines on or off
Distance along line	The distance between labelling points along each isoline (starting at the middle of each polyline of each isoline) The initial value of the distance value is calculated based on the median extent of the component modelled (1/3 of it).
Proximity	The minimum distance between any two labelling points.
Font size [pts]	Size of letters used for labels. The actual size scales with the height of the 3d view.
Decimal digits	Number of decimal digits after decimal comma to show on isoline labels.

Remark: In general it is advisable to keep the *distance along line* at 1/3 of the component's extent. The *proximity* is commonly changed together with the *font size*. Generally the *Proximity* is always smaller than *Distance along line*.

Remark: Visibility of each separate label is checked against all opaque surfaces and objects within the view. Transparent surfaces and do not obscure labels.

Remark: Labels of [isolines](#) shown with active [isoline option "Bold n-th"](#) are shown on emphasized isolines only.

Isoline Labels

Show labels

Distance along line:

Proximity:

Font Size [pts]

Decimal digits

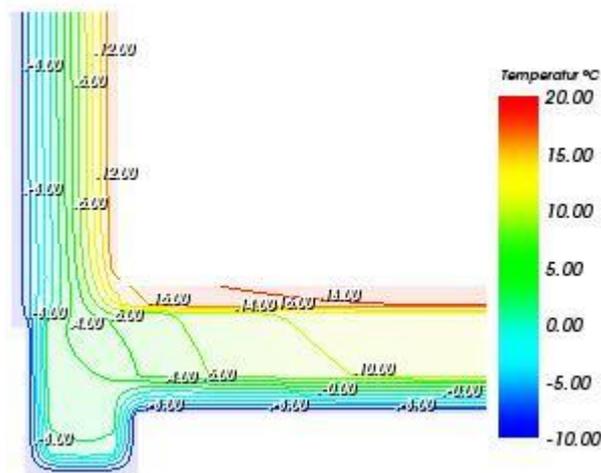
Colorbar Labels

Show labels

intervals

Decimal digits

Labels
Isosurface
Stream
Colorbar
Probe
Axes
3-D



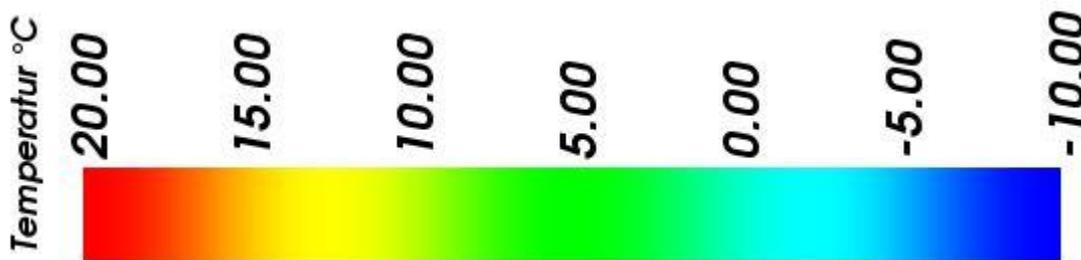
The tricky algorithm positioning the labels can be described as follows:

1. Each isoline is represented by a polyline - under certain circumstances there will be many polylines for the same isoline value generated (sections of isolines).
2. The initial labelling point is set at the middle of such polyline (very short polylines are skipped).
3. Starting from that initial point, in both directions of the polyline, further points will be placed - repeatedly at each *distance along line*.
4. After all points are set for all polylines the distance of each pair of points is tested against *proximity* - one of the points is removed if the distance is less than the set proximity value. This avoids overlapping of labels placed too nearby (below the value of proximity) to each other.
5. All labelling points are tested for their visibility - this highly depends on the orientation of the 3d view. Points obscured by opaque surfaces are removed.
6. All points still on the list are used to place labels in requested *font size*. The size and orientation of labels is independent of the current 3d transformation.

Labelling the Colorbar

Value labels of the [colorbar](#) are placed at one side of the coloured bar.

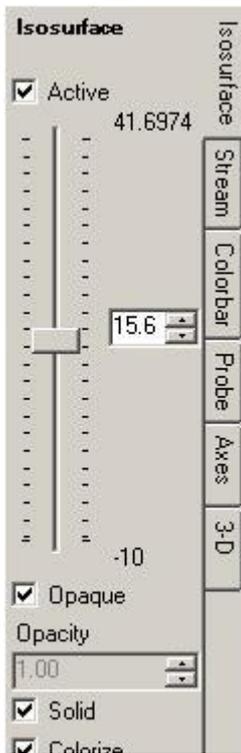
Show labels	Turns labelling of colorbar on or off
# Intervals	Defines the number of intervals at which the colorbar shall be labelled.
Decimal digits	Number of decimal digits after decimal comma to show on colorbar labels..



Remark: When Results3D is showing a periodic evaluation at some time point, the title of the colour bar will display that value.

See also: [Isolines / Isotherms \(control panel\)](#), [Slice X,Y,Z \(control panel\)](#), [Results 3D window](#), [Colorbar \(control panel\)](#)

Isosurface (control panel)



The control panel *Isosurface* is used to adjust parameters of a isosurface - a surface joining points of equal values. The *isosurface* shows a colored surface of constant value of current [primary function](#) (temperature, heat flux etc.).

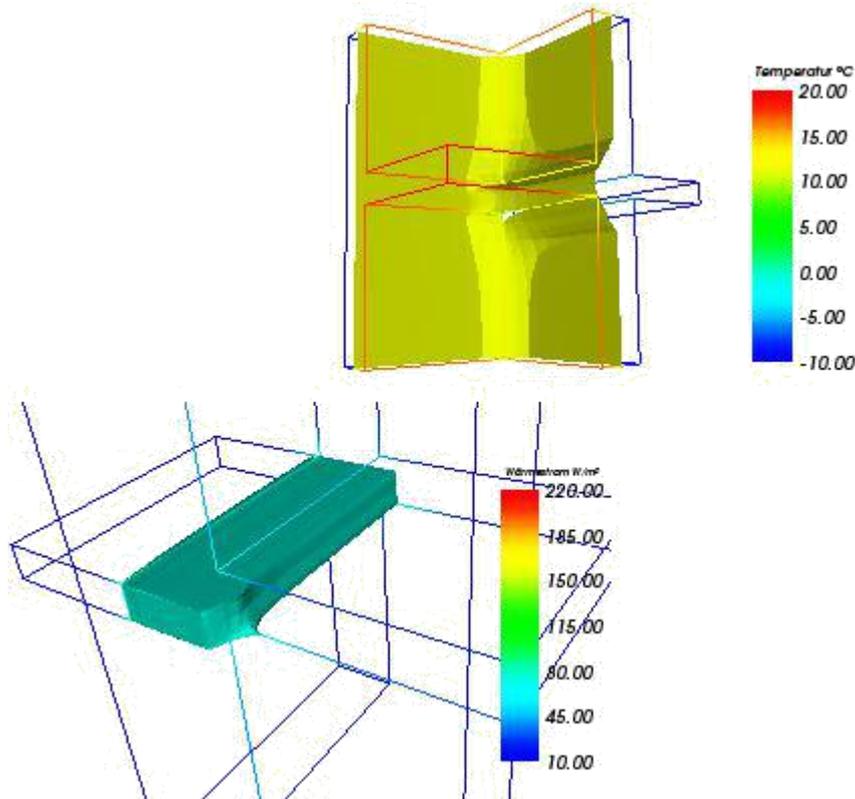
The isosurface is shown [colorized](#). Actual colorizing follows the values of currently [active function](#) (temperature, heat flux, etc. - selected within [General control panel](#)) and currently selected [Colour table](#).

Active, Opaque, Colorize	Decide about the visibility, colorizing and transparency of the surface shown.
Value	The value at which the isosurface shall be generated. The input can be performed either by entry of numeric value in the input field or by moving the slider between the extreme values of respective function.

Remark: Because the isosurface exists within model's interior only it is advisable to either turn the view of the [component's surface](#) completely off or show it partially transparent (i.e. [not opaque](#)) or show it as [wireframe](#) only.

The tooltip of the tab will show up if the [surface](#) obscures objects rendered within the interior of the model.

Remark: Extreme values of temperatures include space temperatures also.



Hint: To identify and locate the are of thermal bridges there is very valuable evaluation offered by visualizing the isosurface of a high value within the field of heat stream densities.

See also: [Results 3D window](#), [Active \(setting\)](#), [Opaque and Opacity \(setting\)](#), [Colorize \(setting\)](#), [Solid or Wireframe \(setting\)](#), [General \(control panel\)](#), [Active Function](#), [Isolines \(Isotherms\)](#)

Streamlines S (control panel)



The control panel *Streamlines* is used to adjust parameters of the stream line evaluation.

A *streamline* shows a trace of a heat flow through any point within the construction. For **2D project** traces are produced through several points placed at the surface of selected space.

If there is vapour solution available **the flow of vapour diffusion** can be visualized with this evaluation alternatively.



The evaluation is performed over the calculated field of **heat flux vectors (or vapour diffusion flux vectors)** at all nodes of the simulation grid – these fields are available only if evaluation of **secondary functions** has been chosen (in **General** the **secondary functions** is active!). If there is no vector field data available to render the tab will show a tooltip stating that.

The streamline will be calculated over the **active vector field** by executing the Runge-Kutta method. The streamline (independently from the **active function** always follows the **active vector field**) is shown **colorized**. Only the colorizing of the streamline corresponds to the **active function** (temperature, heat flux etc.) as set in the **General control panel** and depends on the **colour table** selected.

Streamline through one probe point (Slice X/Y/Z)

The position of the starting point to trace the stream is defined by the intersection of the three **slice planes Slice-X / Slice-Y / Slice-Z**. **Impressive** visualizations can be obtained also by **interactively moving** or **picking the starting point**.

Therefore the **slices X,Y,Z** provide crucial part of parameters required for this evaluation (but need not be visible nor active).

Streamlines at equidistant intervals of heat stream from space boundary in 2D case

For **2-dimensional models** the streamlines can be automatically drawn out of space boundaries at equal intervals of heat stream (of that space). This functionality is available for **2D projects** only.

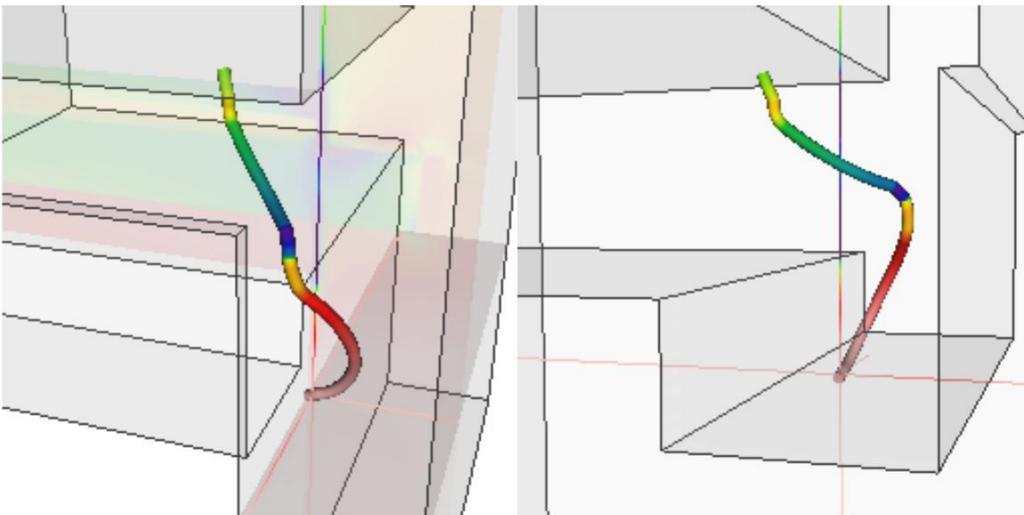
Important: Streamlines are drawn in the **plane of slice Z** which must be active (even if fully transparent)!

After the option "Start at ... Space Boundary" is selected, the name of the space the streamlines shall be drawn from can be chosen together with the number of intervals the stream of that space boundary shall be divided into. The application calculates the heat (or vapour) stream from/to the space, subdivides it into the given number of equal intervals and draws streamlines at each such calculated start value of the stream function.

This evaluation can be executed for 2D projects or for other project types, but only if the construction is absolutely homogenous in the Z-direction and there are no space connections in this direction also. If this variant of evaluation is actually available can be verified within the streamlines control panel - if the choice option "Start at: Space Boundaries" is inactive (shown greyed) then that type of evaluation is not available in the context of actual simulation results ergo of the actual model.

Active , Opaque , Colorize	Decide about the visibility , colorizing and transparency of the streamline
Integration Step length	The factor used to determine the minimal step length of the Runge-Kutta method. Small values lead to high number of iteration steps within each fine grid cell executed by Runge-Kutta method. More such steps (i.e. at small values of integration step) result in smother, more precise, stream lines. Small number of steps (i.e. large integration steps) result in coarse steam flow representation. The value also determines the minimum distance of how close the streamline can approach the surface boundaries of construction. Default: 0.1 mm
Max. propagation length	This number affects the number of integration steps of the Runge-Kutta method performed from the starting point until iterations, and thus tracking the stream field, are aborted. This number is important by defining the highest allowed total length of each separate streamline in millimetres. High values of maximum propagation length will obviously increase the computational time required to create the streamline.

Terminal stream density	This number defines the magnitude of stream density (modulus of flux vector) below which the tracking of the stream field by Runge-Kutta method shall be aborted. Default: 0
Tube Radius	The streamline is shown as a tube to better emphasize the stream (compared to single pixel thick line). The radius is entered in units of the coordinate system - i.e. millimetres. Setting the tube radius = 0 will show it as simple line. Default: The initial value of tube radius is computed based on median extent of component (1/200 of it). Remark: The number of sides rendered for the tube can be adjusted within application settings.



Start at: <ul style="list-style-type: none"> • ProbeX/Y/Z • Space Boundary 	The position of starting point is defined by the intersection of the three slice planes Slice-X / Slice-Y / Slice-Z or, for 2 dimensional models , one can chose to automatically generate starting points on space boundaries at equidistant intervals of heat stream. Remark: The "Start at Space Boundary" option is available for 2D projects only.
Space Boundary	The name of the space from which the streamlines shall be drawn
# intervals	Number of intervals the stream function calculated at the space boundary shall be divided into thus determining the starting points for streamline generation. Remark: High number of streamlines generated will result in high graphical memory demand. It is advised to set the tube radius to 0. Remark: If the stream flows to the space and (at other section of the space boundary) from the space, the highest amount is taken and divided into intervals. To visualize the dividing line between the areas of stream loss and stream win select division at 1 interval only.

Remark: Because streamline (the flow) exists in model's interior only it is advisable to either turn the view of the [surface](#) completely off or show it partially transparent (i.e. [not opaque](#)) or show it as [wireframe](#) only. The same applies to the visibility of [slice planes X, Y, Z](#) (starting point of stream tracing) which shall be made [partly transparent or even completely invisible \(Opacity = 0\)](#).

The tooltip of the tab will show up if the [surface](#) obscures objects rendered within the interior of the model.

Remark: When [surface humidity](#) is selected as active function (panel [General](#)), the streamline is coloured grey only, because the [function of surface humidity](#) is defined at [component's surface](#) only.

Remark: Current implementation of the solution of the initial boundary problem posed here uses Runge-Kutta method of

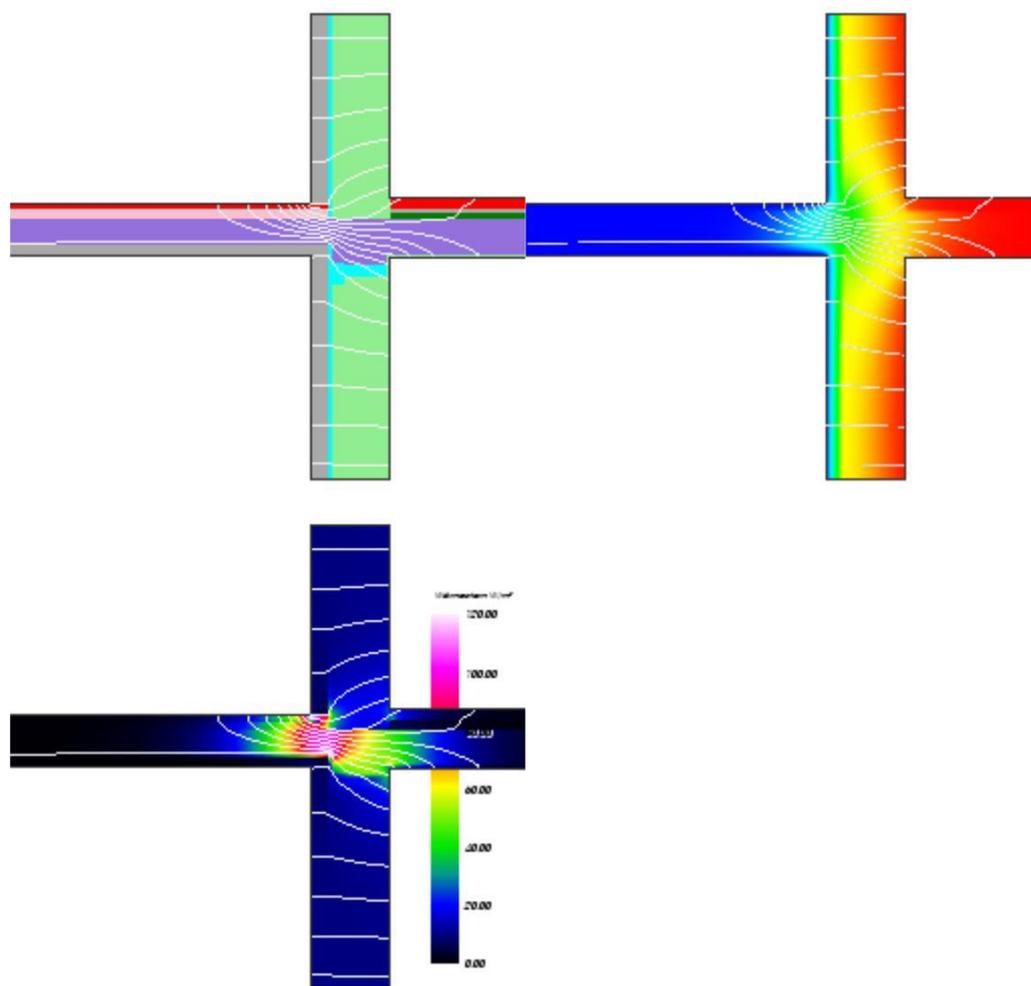
fourth order with adaptive step control. The boundary condition of step control is set to the small, still numerically stable, error value of $\epsilon = 10^{-17}$ and cannot be changed by the user.

The initial-, minimal- and maximal integration steps are determined by the user ([Integration Step length](#)). The minimal step corresponds to the value entered (default 0.1 mm). The initial step length is 100 times the minimal value. The maximal step length is 10000 times the minimal value.

The [maximal propagation length](#) corresponds to the total length of the resulting streamline in millimetres.

Remark: Not to overwhelm the capacity of today's hardware equipment when combined with tube visualization following restrictions apply and are automatically checked and set:

- Showing the streamline as [Tube](#) will be suspended (and all streamlines shown only as simple lines) when for all streamlines taken together the overall number of vertices needed for the tube representation would exceed 1.6 millions (i.e. number of segments $< 1.6\text{Mio}/\text{NumberOfTubeSides}$).
- Rendering as tube will be suspended for all streamlines also if any part of any streamline themselves would contain more than 100.000 segments - such too long streamline, then shown by simple lines, will show broken early. This applies to both parts (the forward and backward) of each separate streamline independently.



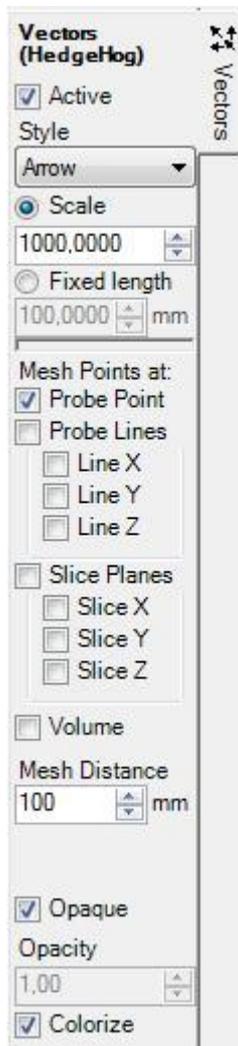
Remark: Start at – Probe X/Y/Z: The position of starting point of a streamline is defined by the intersection of the three [slice planes Slice-X / Slice-Y / Slice-Z](#). The intersection of the three planes must be within model's interior.

If the intersection of the slices X/Y/Z (we call it the probe point) is located very close to components surface it might happen that no stream line can be started from there. Shall this (rare) situation arise you will have to shift the starting point by at least the half of the smallest size of the fine grid cells (i.e. the [start thickness of the fine grid parameters](#), typically 0.5mm or 1mm).

Remark: Start at – Space Boundary: Produces many streamlines started at selected space boundary. The application calculates the sum of stream flowing to the space and the sum of stream leaving the space through the boundary (for exterior space, case with two spaces without powers sources one of this values is always 0). The sum with the largest modulus is divided into the number of intervals entered (maximum 200 intervals, i.e. at 0.5% of the heat stream). Along the space boundary the application repeats the calculation of the stream function and places start points for the streamline at interval boundary values calculated earlier (interpolated). To ensure, that every streamline will start each starting point is

shifted towards the interior of the model by about the half thickness of the surface fine grind cell (typically 0.5mm or 1mm). See also: [Results 3D window](#), [Active \(settings\)](#), [Opaque and Opacity \(setting\)](#), [Colorize \(setting\)](#), [Solid or Wireframe \(setting\)](#), [General \(control panel\)](#), [Active Function](#), [Isolines \(Isotherms\)](#), [Slice X,Y,Z \(control panel\)](#), [Surface \(control panel\)](#), [Vectors \(HedgeHog, Arrows\) \(control panel\)](#)

Vectors (HedgeHog, Arrows) (control panel)



The control panel *Vectors (HedgeHog)* is used to adjust parameters of the vector field evaluation with arrow symbols.

Arrow symbols shows the direction of heat flow through within the construction at any arbitrary point or points on 1,2 or 3-dimensional regular mesh.

If there is vapour solution available [the flow of vapour diffusion](#) can be visualized with this evaluation alternatively.



The evaluation is performed over the calculated field of [heat flux vectors \(or vapour diffusion flux vectors\)](#) at all nodes of the simulation grid – these fields are available only if evaluation of [secondary functions](#) has been chosen (in [General](#) the [secondary functions](#) is active!). If there is no vector field data available to render the tab will show a tooltip stating that.

Vector glyphs will be calculated over the [active vector field](#). Vectors (which direction and eventually size independently from the [active function](#) always follow the [active vector field](#)) are shown [colored](#). Only the coloring of vector glyphs corresponds to the [active function](#) (temperature, heat flux etc.) as set in the [General control panel](#) and depends on the [colour table](#) selected.

The length (and thickness too) of arrow glyphs follow the magnitude (vector norm) the [active vector field](#) (Scale) or are shown at fix constant size.

Alternatively to showing arrow glyphs cone symbols or lines can be chosen.

Arrows at one probe point (Slice X/Y/Z) or Mesh Points

The position of the point to display the arrow glyph is defined by the intersection of the three [slice planes](#) [Slice-X / Slice-Y / Slice-Z](#).

Therefore the [slices X,Y,Z](#) provide crucial part of parameters required for this evaluation (but need not be [active](#) nor visible).

Starting from that probe point a regular mesh of points can be created (at given step, distance) and that either along the Line X,Y or Z, the slice plane X,Y or Z or throughout the whole volume.

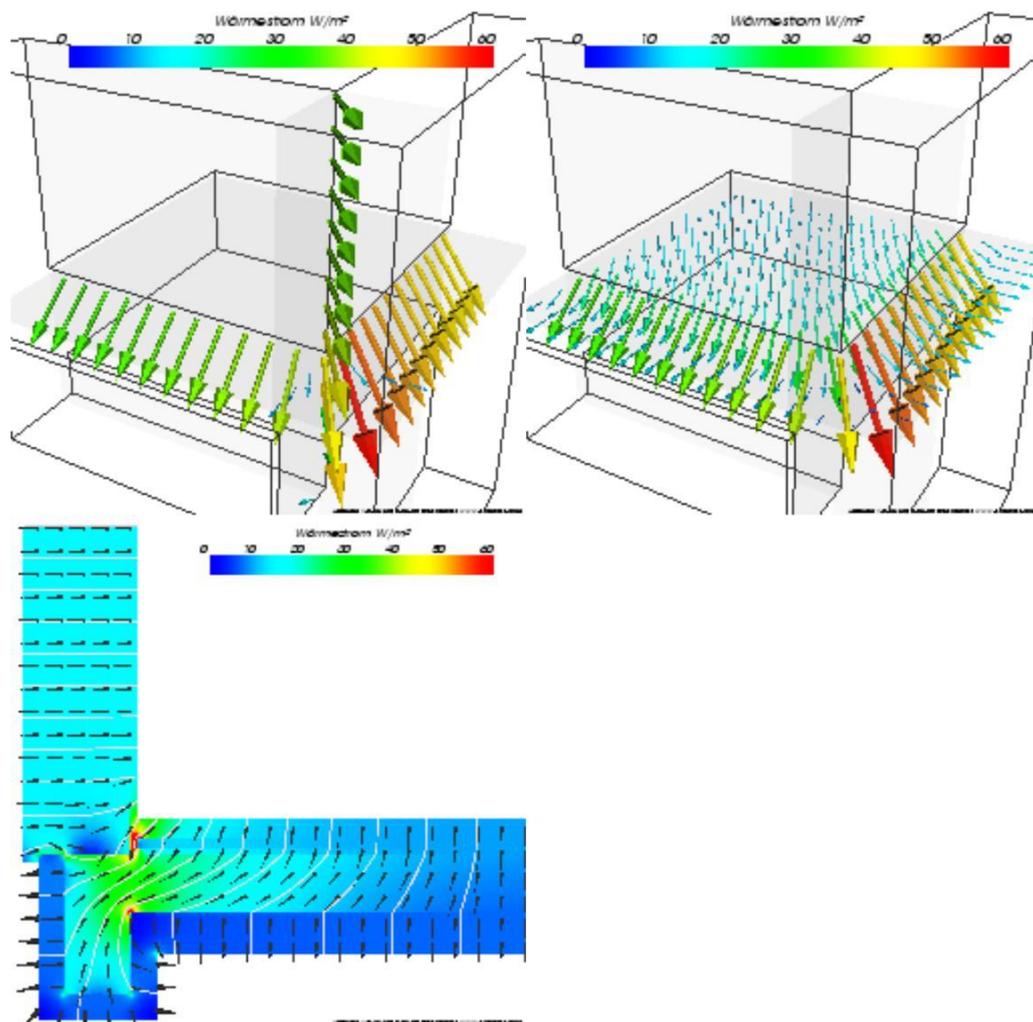
Active , Opaque , Colorize	Decide about the visibility , colorizing and transparency of glyphs
Style	The representation of an glyph: <ul style="list-style-type: none"> • Arrow, • Smooth-Arrow, • Cone, • Smooth-Cone or • Line. Remark: Rendering details of cone glyphs are set within application settings. Remark: Rendering details of arrow glyphs are set within application settings.
Scale	The size (length) of the glyph shall scale by magnitude (vector norm) the active vector field . The normalized magnitude is then scaled by the factor given. Remark: Scaling the normalized magnitude means, that the maximum value within the vector magnitude field is 1.0.
Fixed length	The size (length) of the glyph shall be exactly the value given (in millimetres).
Mesh points at: <ul style="list-style-type: none"> • Probe Point • Probe Lines (LineX,Y,Z) • Slice Planes 	The position of vector glyphs is defined either by the intersection of the three slice planes Slice-X / Slice-Y / Slice-Z or at a regular mesh of points along the LineX,Y or Z, the slice plane X,Y or Z or throughout the whole volume. Remark: The setting "Volume" in combination with too dense mesh might result in display of too much information to be interpreted.

<ul style="list-style-type: none"> (SliceX,Y, • Volume 	
Mesh Distance	The name of the space from which the streamlines shall be drawn Remark: The resulting number of mesh points is limited by following criteria: 500 intervals at each axis direction; total maximum 100000 points

Remark: When [surface humidity](#) is selected as active function (panel [General](#)), glyphs are coloured grey only, because the [function of surface humidity](#) is defined at [component's surface](#) only.

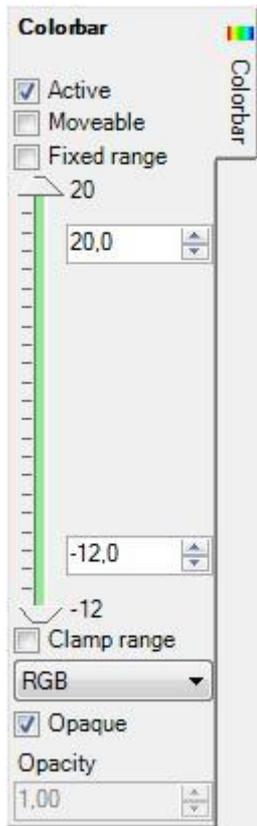
Remark: Because starting points for the glyphs exist within model's interior only it is advisable to either turn the view of the [component's surface](#) completely off or show it partially transparent (i.e. [not opaque](#)) or show it as [wireframe](#) only. The tooltip of the tab will show up if the [surface](#) obscures objects rendered within the interior of the model.

Remark: Glyphs will not be placed within regions of stagnation (flux <10e-17).



See also: [Results 3D window](#), [Active \(settings\)](#), [Opaque and Opacity \(setting\)](#), [Colorize \(setting\)](#), [Solid or Wireframe \(setting\)](#), [General \(control panel\)](#), [Active Function](#), [Isolines \(Isotherms\)](#), [Slice X,Y,Z \(control panel\)](#), [Surface \(control panel\)](#), [Streamlines \(control panel\)](#)

Colorbar (control panel)



The control panel *Colorbar* defines transformation settings of the [colour table](#) used for false colour assignments in all [colorized](#) evaluations (the transformation from function value to a colour). The display of the *colour scale* can be adjusted here also.

The colour table "affects" the [active function](#) (temperature, heat flux, surface humidity, etc.) - selected within the [General control panel](#).

Remark: By changing the setting [Colorize](#) of each respective evaluation the function of coloring by a colour table can be turned off selectively - objects are then homogenously coloured by dark or bright colour instead of being colored by value.

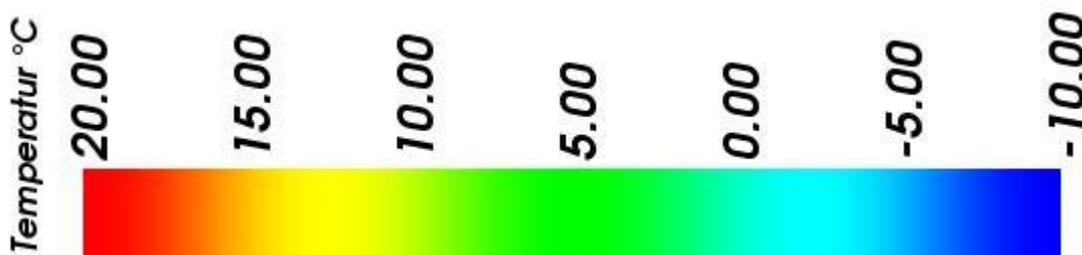
Active, Opaque	Decide about the visibility and transparency of the colour scale and its labels. Remark: Even if the colour scale itself is inactive (not visible) chosen parameters are applied to evaluations!!
Moveable	If turned off all mouse events will be directed to the 3D evaluation - the colour bar cannot be sized nor moved anymore.
Fixed range	If turned on the value range for adjusting the colour bar can be set arbitrarily, independent of the current value range of the evaluated function.
Range-Min/Max	The values of Maximum and Minimum, shown at the top and bottom of the panel, display the current extent of selected evaluation function. When " Fixed range " is chosen both values can be set arbitrarily, independent of the current value range of the evaluated function.
Min/Max	The mapping interval of the colour bar displayed can be reduced by the input in the respective fields. The input can be performed either by the entry of numeric values into the respective input field or by moving the sliders between the extent values. This allows better emphasizing of selective value interval.
Clamp range	Allows for clamping of evaluation areas outside of selected value interval. Remark: Current implementation provides very simple but extremely fast clamping function.
Colour table	Preset with the standard value "(Default)" the colour table of red through green to blue is used. The application provides many various colour tables which can be selected here.

Labelling of the colour bar can be adjusted within the [control panel Labels](#).

The [position at which the colour bar is displayed](#) can be arbitrarily changed within the 3D view by moving and sizing it with the mouse.

Remark: Even if the colour scale itself is [inactive](#) (not visible) chosen coloring parameters are applied to evaluations!

Remark: Extreme values of temperatures include space temperatures also.



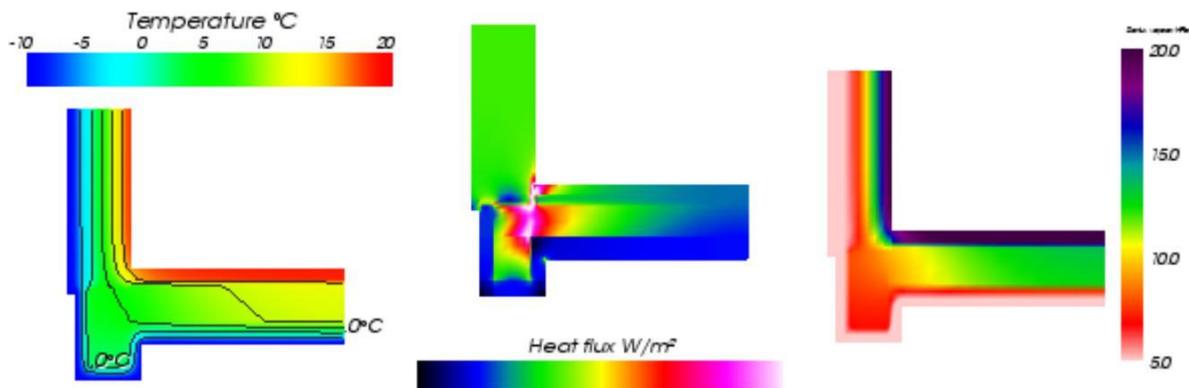
Hint: [Fix the Range of the Color Bar](#) to receive comparable results by decoupling it from the actually evaluated domain. This [function specific setting](#) takes care for keeping [the range constant](#) regardless of changes to [boundary conditions](#), [evaluation time point](#), [animation](#) or for different projects.

Position and size of the colour bar display

The position at which the colour bar is displayed can be arbitrarily changed within the 3D view by moving and sizing it with the mouse if the "[Moveable](#)" setting is turned on.

You interactively manipulate the position, size, and orientation of a colour bar. It reacts to left mouse events and mouse movement. The shape of the cursor will change based on its location. If the cursor is over an edge of the colour bar the cursor shape will change to a resize edge shape.

If the position of a colour bar is moved to be close to the centre of one of the four edges of the 3D view, then the colour bar will change its orientation to align with that edge. This orientation is sticky in that it will stay at that orientation until the position is moved close to the centre of another edge.



Actual position of the colour bar is saved automatically within [application settings file](#) and restored the next time the [Results3D window](#) is opened.

Remark: If the case the colour bar has been accidentally moved outside of the visible area of the window, it can not be picked with the mouse pointer anymore for any further movement. To restore the position to some position where it can be picked and moved again you must close the 3D window and reopen it - the colour bar will be automatically restored at some visible position nearby the image edge.

Remark: When Results3D is showing a [periodic evaluation at some evaluation time point](#), the title of the colour bar will display that value (lacking other annotations the title of the colour bar has been chosen for now as appropriate).

<Function name><Unit> (<Time Point>)

For vertically displayed colour bar the time information will be placed onto the second line of the title.

TRANSIENT-option: Solving and evaluating time dependant dynamic, periodic problems when heat capacity effects are concerned is only possible with an active TRANSIENT-Option of the program.



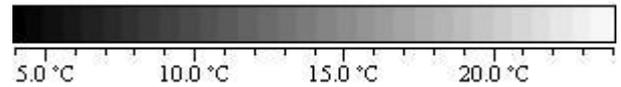
See also: [Colorize \(setting\)](#), [Results 3D window](#), [Isolines / Isotherms \(control panel\)](#), [Surface \(control panel\)](#), [Slice X,Y,Z \(control panel\)](#), [Streamlines \(control panel\)](#), [Vectors \(HedgeHog, Arrows\) \(control panel\)](#), [Isosurface \(control panel\)](#), [General \(control panel\)](#), [Labels \(control panel\)](#), [Fixed Range \(setting\)](#)

Colour scale / Colour tables

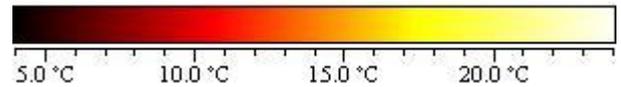
The application provides various colour scales (colour tables) used to assign colours to numerical results within graphical evaluations. Particular selection of a colour scale depends on visualisation needs of the user.

In addition to the obligatory RGB and RGB_INV tables, following colour tables are available:

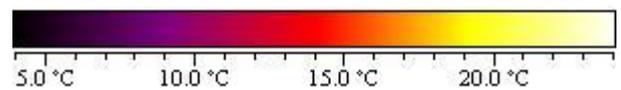
The **grey scale** provides the most simple palette of colours. Temperature values are transformed to brightness values, similarly as it is done by camera. This palette is useful for exceptionally neutral evaluations.



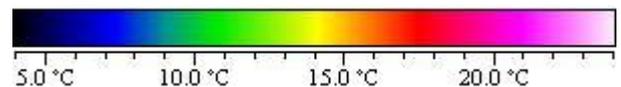
Especially for temperature images it is natural to use the colour scale of **glowing body**: Cold objects display dark and reddish, hot show in bright in yellow or even white. The contrast is moderate, but higher than of the neutral grey scale.



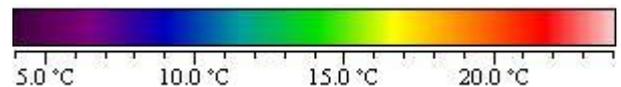
The contrast of a thermogram can be enhanced by adding the colour of violet between red and the black (= cold) end.



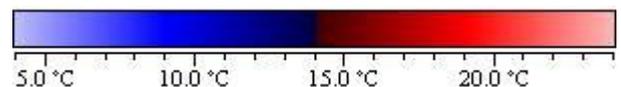
Alternatively a thermogram can be enhanced also by using the contrast of a scale of hot stars as it is used by **astronomy**.



To carry the contrast to extreme by adding as many colours to the scale as possible. One can use **psychological perception** people see as "cold" and "warm" colours, and assign them accordingly to temperatures. None of colours used is repeated to avoid any ambiguity. It is worth to observe, that even small value errors will be visible now.



Cartography uses an alternative kind of colouring allowing perception of even very small value differences. Psychologically "colder" colours are assigned to smaller values and "warmer" colours apply to higher values.

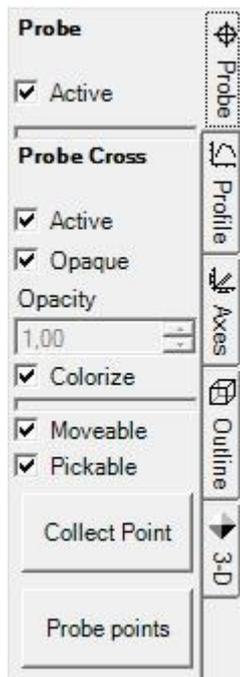


There are some use cases making conventional coloring not suitable - for example when the picture of a thermogram shall allow derivation of energy **balance**. For such situations a symmetrical colouring is more reasonable and the centre jump point could be set to zero.

Remark: The current number of colour intervals used within the colour table is 2048 (more colours „cannot be seen simultaneously by human eye“).

See also: [3D evaluations](#)

Probe (control panel)



The control panel *Probe* is used to control the view of [data measurement](#) at some probe point, [graphical display](#) of its location and control of additional [interactive positioning](#) methods.

The position of the probing point is defined by the intersection of the three [slice planes Slice-X / Slice-Y / Slice-Z](#). [Here](#) it can be interactively moved by [dragging the 3D cursor](#) with the mouse or [by picking points](#) with the mouse at any displayed surface.

Probe	
Active	Decides about the visibility of the probe data for its textual readout .
Probe Cross	
Active, Opaque, Colorize	Decide about the visibility , colorizing and transparency of the probe cross
Moveable	
	When active allows interactive moving of the 3D cursor (by dragging it with the mouse) and thus arbitrary placement of the probe point position X,Y,Z.
Pickable	
	When active allows picking the probe point position at any surface object displayed by clicking (or double clicking) on it.
Collect Point	
	Coordinates X,Y,Z of the current probe point are added to the list of probe points (this list will be edited within the Probe Points window and evaluated within Probe Points part of Results report). The window Probe Points will be exposed also.
Probe points	
	The window Probe Points will be exposed.

Readout of the probe point data

The result of this probing evaluation is shown at the bottom left corner of the view. The colour for that display is chosen automatically and set to some high contrast colour against current [background colour](#).

At the evaluated point its coordinates are output (the values of X,Y and Z) along with the calculated value of temperature. If required tri-linear interpolation is used to calculate exact resultant value at any point of modelled construction from values calculated by the simulation at nodes of the [super fine grid](#).

The probe provides the way to read out a value (e.g. temperature in °C) at any point of the construction.

If the probing point is placed at space boundary the value of non condensing (maximum) air humidity (%) is evaluated and output also.

If there is solution of vapour diffusion available also, probing shows the pressure difference of partial and saturation pressures (hPa) too.

If the point is positioned at the exterior of modelled construction no results are output beyond coordinate data.

Remark: The [Probe Points part of Results report](#) provides an alternative textual method of evaluating temperature values at multiple [probe points](#) .

Visualizing the probe point location

At the intersection lines of slice planes X,Y,Z a [colored](#) probe cross can be displayed. Actual colorizing follows the values of [active function](#) (temperature, heat flux (stream density), etc. - selected within [General control panel](#)) and currently selected [Colour table](#).

Remark: Because this evaluation provides reasonable results for points located in model's interior only it is advisable to either turn the view of the [surface](#) completely off or show it partially transparent (i.e. [not opaque](#)) or show it as [wireframe](#) only.

The tooltip of the tab will show up if the [surface](#) obscures objects rendered within the interior of the model.

Moving the probe point interactively (3D cursor moveable)

Moveable The setting *Moveable* controls the 3D cursor. The setting can have one of the three distinct states:

Checked	<i>Immediate</i>	3D cursor is active. While the 3D cursor is being dragged with the mouse its position will be immediately applied to the values of Slice X,Y,Z thus resulting in frequent (interactive) updates of the rendering.
Partly checked	<i>Deferred</i>	3D cursor is active. The new position of the 3D cursor will be passed to Slice X,Y,Z coordinates at the end of the move (i.e. when the mouse button is depressed). This ensures that the eventually complex visualisation pipeline is executed at final position once only.
Unchecked	<i>Off</i>	Using the 3D cursor is disabled.

Interaction with the 3D cursor will start as soon as the mouse is positioned in the neighbourhood of the probe cross or rectangle lines surrounding the slice planes (so called X/Y/Z shadows) and the left mouse button is depressed. Elements of the 3D cursor will display in light green during the active interaction.

If the shift key is held depressed during the initial move of the 3D cursor, then that initial move direction constrains the further interaction to one axis direction only.

Remarks: The widget itself is shown nearly transparent not to overlap with other elements (opacity = 0.1). The outline shadow (the box) is not shown by design. When selected default colour used (light-green). Scaling/Zooming/Translation of the 3D cursor outline is not allowed by design. Movements are always constrained to the bounding box of the model.

Remark: During [export operations](#) the 3D cursor widget will be hidden temporarily and not rendered to the resulting export view.

Picking the probe point at surfaces shown (pickable)

Pickable The setting *Pickable* controls the picking behaviour of the mouse. The setting can have one of the three distinct states:

Checked	<i>Double Click</i>	Picking a point on some visible surface is initiated with a double click.
Partly checked	<i>Single Click</i>	Picking a point on some visible surface is initiated with one single click.
Unchecked	<i>Off</i>	Picking is disabled.

Upon the click (or double click) the application will attempt to pick the point on the next visible surface below the current cursor position.

The picking of the point is available at visible (i.e. active, opaque or partly transparent) surface objects and provided in that order:

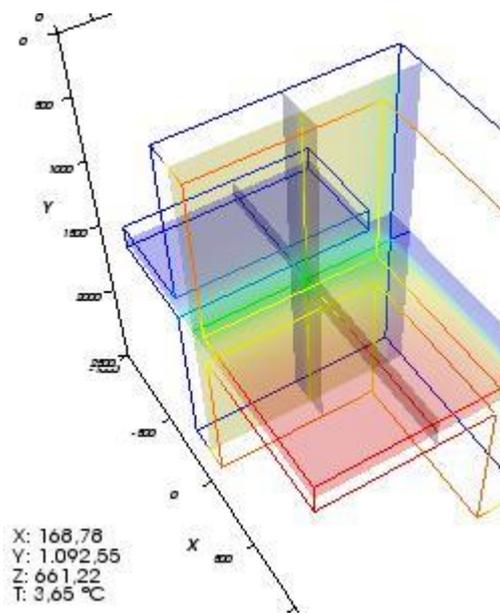
- [Surface](#) - locating probe output at interesting places of the analyzed scalar function.
- [Isosurface](#) - usable to pick a starting point of [streamline](#) on the isosurface of high heat flux (identifying thermal bridges).
- [Slices X,Y,Z](#) - usable to re-position two slice planes at some location at the third plane

Remark: if neither surface object is visible (inactive or totally transparent) picking will have no effect.

Remark: [The position of SliceX/Y/Z](#) will be retained within application settings and restored respectively.

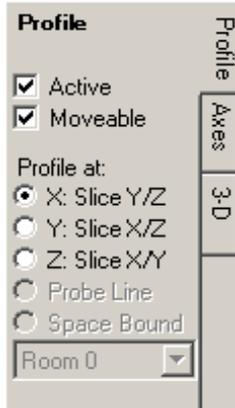
If the old slice position is outside of the model bounds (with model spatially changed or different model loaded) the respective slice position will be reset to the centre of the model. See also [Application setting "Restore last Result3D Parameters"](#).

Remark: From the [Probe points window](#) you can double click onto a data mark record of some point listed there to transfer its coordinates to the 3D evaluation (position of [slices X/Y/Z](#)).



See also: [Results 3D window](#), [Active \(setting\)](#), [General \(control panel\)](#), [Active Function](#), [Slice X,Y,Z \(control panel\)](#), [Surface \(control panel\)](#), [Profile \(control panel\)](#), [Probe Points window](#)

Profile (control panel)



The control panel *Profile* defines parameters for the visualization of values of the [active function](#) along specified line or edge showing within X/Y-diagram.

Active	Decide about the visibility of the function diagram.
Moveable	If turned off all mouse events will be directed to the 3D evaluation - the diagram cannot be sized nor moved anymore.
Profile at: X:SliceY/Z Y:SliceX/Z Z:SliceX/Y	Selects the line used to profile the values of the active function . The location of the line is set by values controlled within panels Slice X, Y and Z .

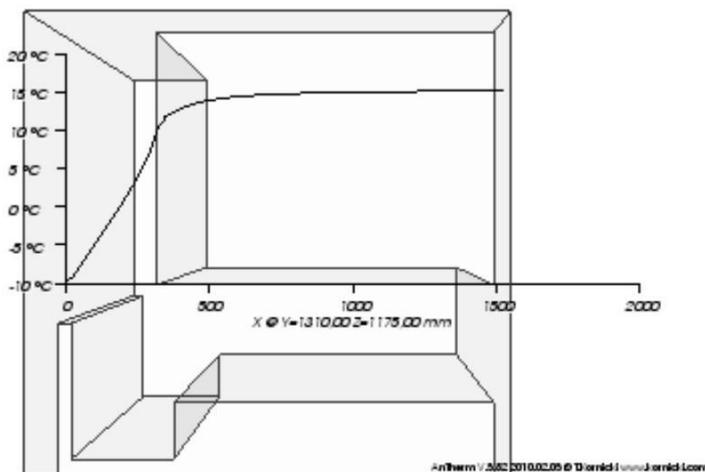
Labels of the horizontal axis display coordinate values along the chosen edge line. The axis title report the exact position and direction of the profile line.

The [position at which the diagram is displayed](#) can be arbitrarily changed within the 3D view by moving and sizing it with the mouse.

Remark: Adjustments of axis labels can be done within [panel Axes](#).

The colour of the function line, axes lines and annotations is chosen automatically and set to some high contrast colour against current [background colour](#).

Remark: Settings "Probe Line" and "Space Boundary" are yet inactive within current implementation.



Position and size of the diagram display

The position at which the diagram is displayed can be arbitrarily changed within the 3D view by moving and sizing it with the mouse if the ["Moveable" setting](#) is turned on.

You interactively manipulate the position, size, and orientation of the diagram. It reacts to left mouse events and mouse movement. The shape of the cursor will change based on its location. If the cursor is over an edge of the diagram the cursor shape will change to a resize edge shape.

If the position of diagram is moved to be close to the centre of one of the four edges of the 3D view, then the diagram will change its orientation to align with that edge. This orientation is sticky in that it will stay at that orientation until the position is moved close to the centre of another edge.

Actual position of the diagram is saved automatically within [application settings file](#) and restored the next time the [Results3D window](#) is opened.

Remark: If the case the diagram has been accidentally moved outside of the visible area of the window, it can not be picked with the mouse pointer anymore for any further movement. To restore the position to some position where it can be picked and moved again you must close the 3D window and reopen it - the diagram will be automatically restored at some visible position nearby the image edge.

See also: [Results 3D window](#), [Slice X, Y, Z \(control panels\)](#), [Axes \(control panel\)](#)

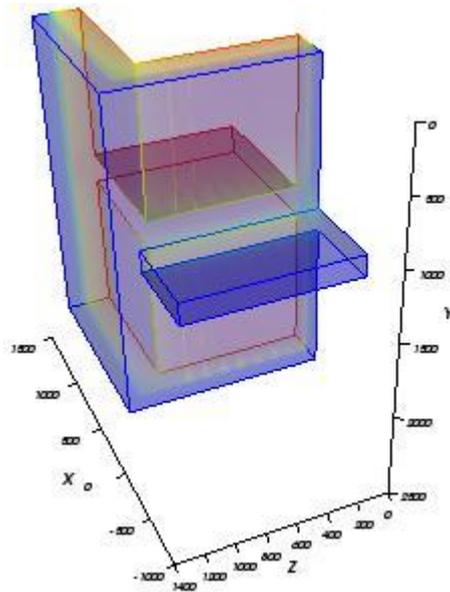
Axes (control panel)

The control panel Axes is used to adjust setting of the axes representation shown around the three dimension view of evaluation combination.

Active, Opaque	Decide about the visibility and transparency of the axes lines and their annotations.
Placement	At Outer Edges - axes will be placed at outer edges of the volume or At Closest Triad - axes will be placed at the corner which is closest to the viewer.
Titles	Decide about the visibility of titles on all three axes (e.g.. X, Y, Z)
Font Size [pts]	Defines the size of the font used to show axes titles.
Labels	Decide about the visibility of numerical labels on axes.
Font Größe	Defines the size of the font used to show numerical labels on axes.
# intervals	Defines the number of intervals of numerical labels on all three axes.
X Aktiv	Decide about the visibility of X axis and its annotations.
X Titel	Decide about the visibility of an axis tile on X axis.
Titel	Defines the title of X axis
X Labels	Decides about the visibility of numerical labels on X axis.
Y Aktiv	Decide about the visibility of Y axis and its annotations.
Y Titel	Decide about the visibility of an axis tile on Y axis.
Titel	Defines the title of Y axis
Y Labels	Decides about the visibility of numerical labels on Y axis.
Z Aktiv	Decide about the visibility of Z axis and its annotations.
Z Titel	Decide about the visibility of an axis tile on Z axis.
Titel	Defines the title of Z axis
Z Labels	Decides about the visibility of numerical labels on Z axis.

By default axes are shown with numerical labels and titles. The colour of axes lines and annotations is chosen automatically and set to some high contrast colour against current [background colour](#).

Axes lines are settled a bit out of the volume shown allowing sufficient space for numerical labels. The position of each axis is adjusted according to the placement chosen (so called "Fly-Mode" method) - for each axis there are 4 different placement alternatives at 4 sides of the [Outline](#) of the volume).



Remark: During 2D evaluations and with views parallel to axes planes (side views) it is advised not to select "At Outer Edges" - otherwise axes will jump between left/right and bottom/top edges because the term "outer" is ambiguous (and numerically unstable thereafter)

Remark: Naming of axes titles can be adjusted in [application settings](#) also.

See also: [Results 3D window](#), [Active \(setting\)](#), [Background colour \(setting\)](#), [3-D Navigation Orientation Marker \(control panel\)](#), [Application settings](#)

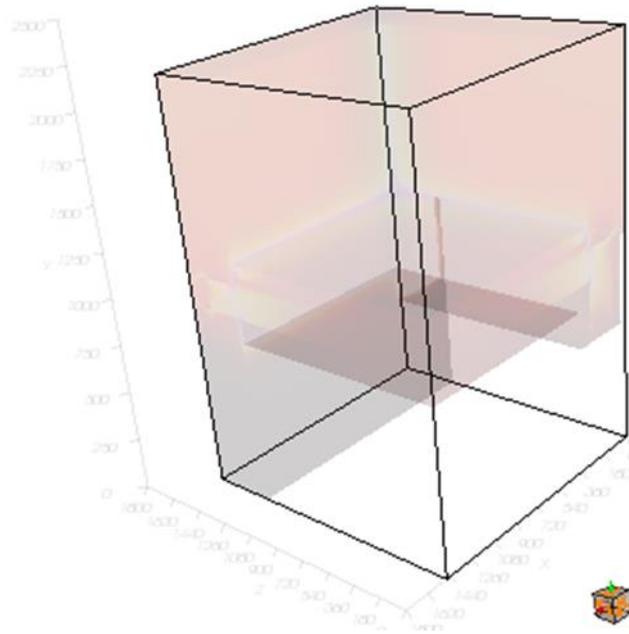
Outline (control panel)



The control panel *Outline* is used to control the display of the bounding box of the visualized object.

Outline

Allows the display of a *bounding-box*: i.e. of a *ashlar* surrounding the whole modelled construction. Only edges of the ashlar are displayed. The outline can be helpful during [positioning the graphical view within virtual 3D space](#), similarly as [axes](#) will do also. The [Active setting](#) turns the visibility of an outline on or off.



See also: [Results3D window](#), [Active \(setting\)](#), [Opaque and Opacity \(setting\)](#), [Surface Edges \(control panel\)](#), [Axes \(control panel\)](#), [3-D Navigation Orientation Marker \(control panel\)](#)

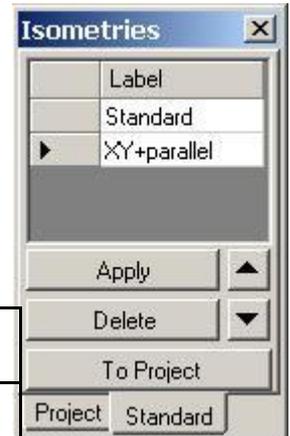
Isometries window

The window *Isometries* shows the list of collected 3D transformations of the 3D view.

Collecting new isometries is initiated from the [Results3D window](#) with the button [Collect Isometries \(in the control panel 3-D\)](#) or from the [View menu](#).

The isometry shown as an item in the list can be easily restored and applied to the 3D view.

The list of isometries specific to the project is shown in the tab **Project**. The tab **Standard** shows the list of [application wide](#) isometry settings.



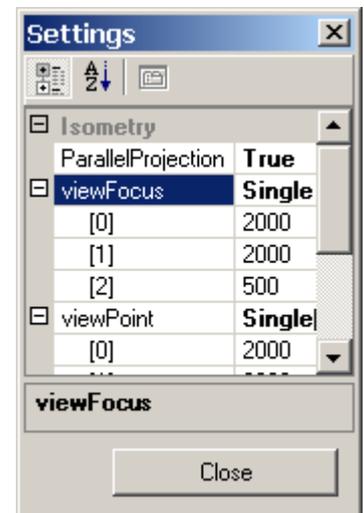
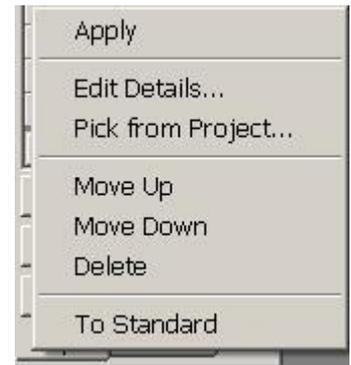
Label	An arbitrary, editable, label of respective item in the list of isometries (used for documentation and better, easier identification only)
Double click onto record mark	Initiates repositioning the current view to the position (3D transformation) clicked within the list. Remark.: Transformation applied is taken from the current row – i.e. even if the click occurred within text of another line, the data applied is taken from the line marked with the record mark (an arrow symbol).
Apply	Initiates repositioning the current view to the position (3D transformation) currently marked within the list.
Delete	Deletes the currently selected (marked) item form the list.
Move Up ▲	The position of the item currently selected can be moved in the list.
Move Down ▼	The position of the item currently selected can be moved in the list.
To Standard	The isometry (3D transformation) associated with the selected entry within the project specific list will be copied (as additional entry) to the application wide list (Standard).
To Project	The isometry (3D transformation) associated with the selected entry within the application wide (Standard) list will be copied (as additional entry) to the project own list.
Projekt	Exposes the list of project specific isometries
Standard	Exposes the list of application wide (standard) isometries

Remark: Hold one of the modifier keys (Alt, Ctrl or Shift) depressed while pressing "Apply" or double-clicking the choice to view animated transformation of the position. The duration of the animation is set via the advanced application setting ["DesiredFlyToSeconds"](#).

Functions of the context menu

The display of the list has a context menu attached (exposed when right mouse button is pressed):

Apply	Initiates repositioning the current view to the position (3D transformation) currently marked within the list.
Edit Details...	Transformation data of the currently selected item can be edited in detail and adjusted according to specific needs. <u>Remark:</u> This is an advanced function. Naming chosen for distinctive parameters shall be self explanatory.
Pick from Project...	The list of project specific isometries saved within some another project will be imported (appended) to the current project's list. <u>Remark:</u> This function is available in the "Project tab" only.
Move Up	The position of the item currently selected can be moved in the list.
Move Down	The position of the item currently selected can be moved in the list.
Delete	Deletes the currently selected (marked) item form the list.
To Standard	The isometry (3D transformation) associated with the selected entry within the project specific list will be copied (as additional entry) to the application wide list (Standard).
To Project	The isometry (3D transformation) associated with the selected entry within the application wide (Standard) list will be copied (as additional entry) to the project own list.



The selected isometry is applied to all 3D windows: [Elements 3D window](#), [Elements 23 window](#), [Results 3D window](#)

Remark: To retain the list of isometries of the project between application executions you must [save the project!](#)

Remark: The application wide list of isometries (Standard) will be [saved with application settings](#) (at application's exit).

See also: [Results 3D window](#), [Elements 23 window](#), [Results 3D window](#), [3-D Navigation \(control panel\)](#), [Settings file AnTherm.exe.Settings.xml](#), [Settings - various \(dialog\)](#)

Active (setting)

Active With the switch *Active* you define if the particular part of evaluation is created and eventually visible.
The setting is available for following [evaluations](#) :

- [Outline](#)
- [Model](#) and [Model Slices](#)
- [Surface](#) and [Surface Edges](#) and [Surface Slice Edges](#)
- [Slice X](#), [Slice Y](#) and [Slice Z](#)
- [Isolines \(Isotherms\)](#)
- [Isosurface](#)
- [Streamlines](#)
- [Vectors \(HedgeHog, Arrows\)](#)
- [Colorbar](#)
- [Probe](#)
- [Profile](#)
- [Axes](#) (also [Axis X](#), [Axis Y](#) and [Axis Z](#))

Remark: Some combinations of evaluations are only possible if some other parts of evaluation are simultaneously active. Such prerequisite dependency examples are:

- [Isotherms](#) on top of a [slice plane X](#) are only visible, if both evaluations, [Isolines](#) and [Slice X](#), are active.
- [Isolines](#) on top of [component's surface](#) - both must be active to receive the visual result.
- etc.

See also: [Results 3D window](#), [Opaque \(setting\)](#), [Solid \(setting\)](#), [Colorize \(setting\)](#)

Opaque and Opacity (setting)

Opaque The setting *Opaque* and the *Opacity* define the level of translucency of graphical objects shown.

Opacity An opaque Object is not translucent. It fully obscures objects located behind it.

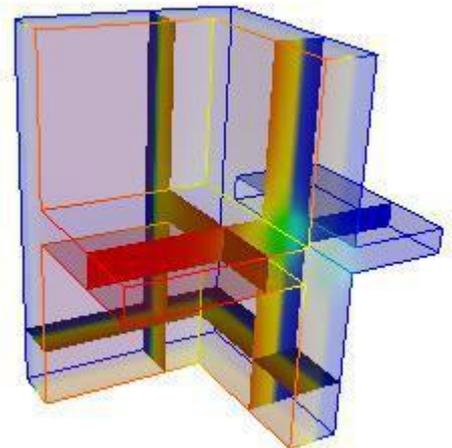
When the switch *Opaque* is turned off, one can set the partial opacity / transparency of the respective object. The value of opacity is in the range of 0 to 1:

- 0 - no opacity, the object is fully transparent (and thus "invisible" also!)
- 1 - full opacity, the object is opaque, obscures and hides all objects "behind" it.

When the value within an interval is chosen (e.g. 0.3) the particular object is still visible, but thanks to its transparency (translucency) does not obscure nor hide object behind it allowing them to be visible also.

The setting is available for following [evaluations](#):

- [Outline](#)
- [Model](#) and [Model Slices](#)
- [Surface](#) and [Surface Edges](#) and [Surface Slice Edges](#)
- [Slice X](#), [Slice Y](#) and [Slice Z](#)
- [Isolines \(Isotherms\)](#)
- [Isosurface](#)
- [Streamlines](#)
- [Vectors \(HedgeHog, Arrows\)](#)
- [Colorbar](#)
- [Axes](#)



Remark: The opacity will be automatically set to 0.1 when the user unchecks the Opaque-switch while the original opacity value was left at 1.0 - this provides transparent visualization immediately after the user requests transparency (but has left the opacity value at 1.0).

See also: [Evaluations](#), [Results 3D window](#), [Active \(setting\)](#), [Solid \(setting\)](#), [More Light](#)

Colorize (setting)

Colorize

The setting *Colorize* define the colorizing of a particular graphical object.

The setting can have one of the three distinct states:

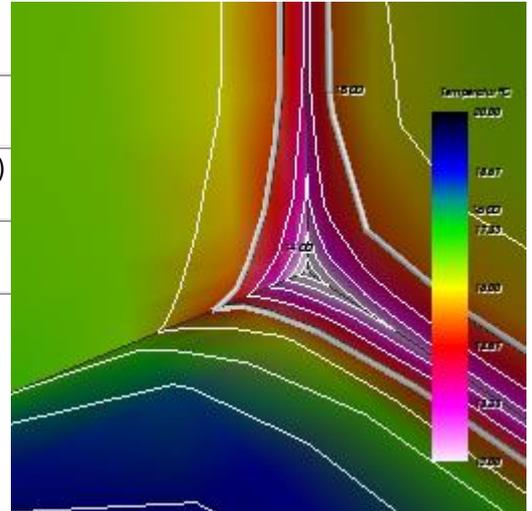
Checked	<i>Colour table</i>	Colorizing follows the currently selected Colour table (or element's properties in model view)
Partly checked	<i>Dark</i>	The object is coloured with some dark (nearly black) colour
Unchecked	<i>Bright</i>	The object is coloured with some bright (nearly white) colour

The setting is available to following [evaluations](#) :

- [Outline](#)
- [Model](#) and [Model Slices](#)
- [Surface](#) and [Surface Edges](#) and [Surface Slice Edges](#)
- [Slice X](#), [Slice Y](#) and [Slice Z](#)
- [Isolines \(Isotherms\)](#)
- [Isosurface](#)
- [Streamlines](#)
- [Vectors \(HedgeHog, Arrows\)](#)

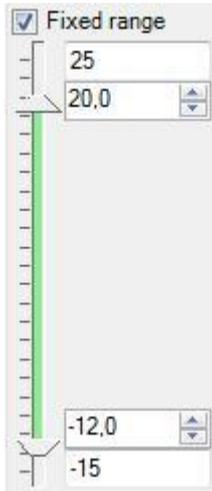
Example: An interesting ability exists when willing to show white isolines over colored (by value) component's surface, while emphasizing component's edges in black.

See also: [Evaluations](#), [Results 3D window](#), [Active \(setting\)](#), [Solid \(setting\)](#), [More Light](#), [Colorbar \(control panel\)](#), [Colour scale / Colour tables](#), [Fixed Range \(setting\)](#)



Fixed Range (setting)

Activating the [switch "Fixed Range"](#) (here referenced in the context [ColorBar](#) by example) allows to override (fix) the current value range of the [active evaluation function](#).



If ["Fixed Range" is turned OFF](#) (default) the overall value range, calculated out of the domain of current values set of the active evaluation function, is displayed at the top and bottom of the panel and is customizable. This domain is subject to automatic adjustment whenever the different [active function](#) is chosen, [boundary conditions](#) or [time point of evaluation](#) are changed or the project itself has been modified.

Because the ["Min/Max" interval boundaries](#) must always stay within this domain, they might be modified automatically too (e.g. if the currently set minimum underflows new data of the active function after some change to the boundary conditions).

When the ["Fixed Range" is ON](#) the [domain range is entered manually](#) (overridden) and remains invariant, independent of currently evaluated data set or the project.

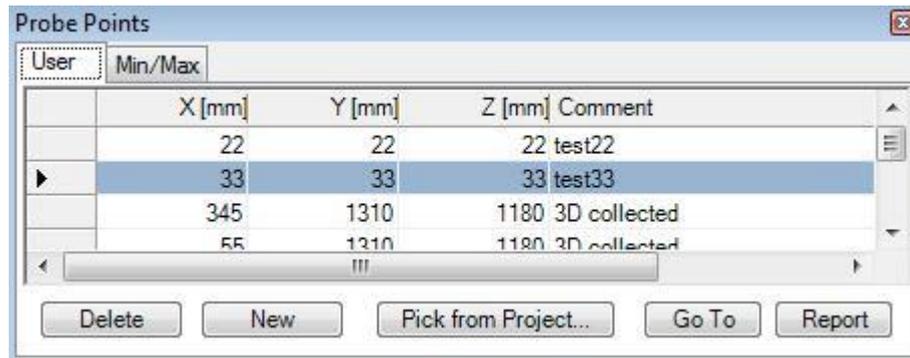
Important: The domain can supplied separately for any [active function](#) and will be retained [as function specific setting](#). By thy the ["Min/Max" interval boundaries](#) will not be varied too.

Remark: With any new boundary conditions the ["Min/Max" interval boundaries](#) will not change (the [domain range](#) will eventually be extended but never it will shrunk).

For example this can keep the setting of colour bar used for evaluation of temperature field independent of the project or [boundary conditions](#) always in $-10/20^{\circ}\text{C}$ and for the purpose of pressure difference in $-5/5\text{hPa}$. In the combination with [function specific](#) choice of colour table this leads to optimal and comparable visualization of results, even for different projects.

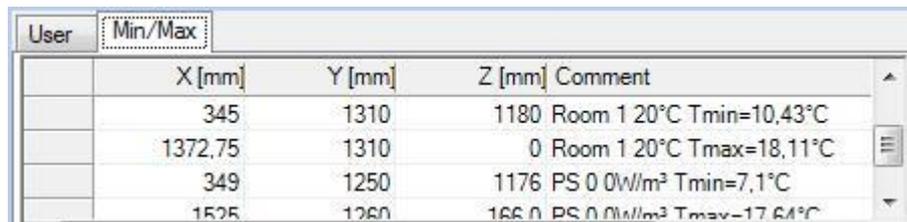
See also: [Colorbar \(control panel\)](#), [Isolines / Isotherms \(control panel\)](#), [General \(control panel\)](#)

Probe Points / Extreme (min/max) Locations window



The window *Probe Points* is used to input any number of arbitrary points for which the [Probe Points report](#) shall be generated (second part of [Results report](#)). The list of probe points can be extended from the [Result 3D window \(Probe\)](#) also (by "Collect point").

The Min/Max tab of the window lists interesting locations with extreme values (e.g. points lowest and highest temperatures at surfaces of all spaces and within the interior of all power sources) thus providing the ability to [select them for further evaluation](#). This read only listing is automatically provided and updated as soon as any [boundary conditions have been applied](#) to the solution.



X, Y, Z	Coordinates of the point in millimetres
Comment	An (optional) description of the point entered (used for documentation only)
Delete/New	Deletes the current data line or append a new one
Pick from Project...	Probe points saved within another project will be read and appended to the list.
Report	Confirms the data entry and initiates the creation of probe points in the Results report .
Double click onto data record mark	Initiates translation of the current probe point shown in the Result3D window to the position shown in the line clicked - it translates Slices X/Y/Z accordingly. Within a 2D-project the Z-slice is set to 500 mm.
Go To	Remark: It is data shown in the currently marked line transferred - this means, that a double click onto column or new row will also result in setting the current probe point to values shown selected by data mark arrow.

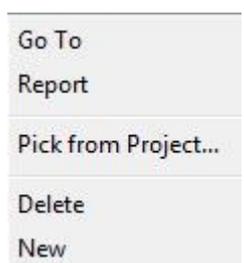
The input is confirmed by pressing the button *Apply*. The [second part of Results report](#) will show evaluation of temperatures for all points (x, y, z coordinates) entered in this window. If some point does not belong to the interior of the construction no temperature values will be shown in the corresponding line of the report.

Probe points entered are saved to the [project file](#).

Remark: Similar [evaluation of one probe point](#) is available within [graphical results](#) of the application.

Remark: The active tab will be saved upon the window is closed together with the application settings (*LastSelectedProbePointsFormTabPageName*) and restored when this window is opened again.

Remark: While ["Collect Point"](#) (e.g. out of ["Results 3D window"](#)) is requested the window will switch to the "User" tab where the new probe point will be appended.



See also: [Results report](#), [Boundary Conditions window](#), [Evaluation windows](#), [Probe \(control panel\)](#), [Coordinate system](#), [Slice X, Y, Z \(control panel\)](#), [Timelines window](#)

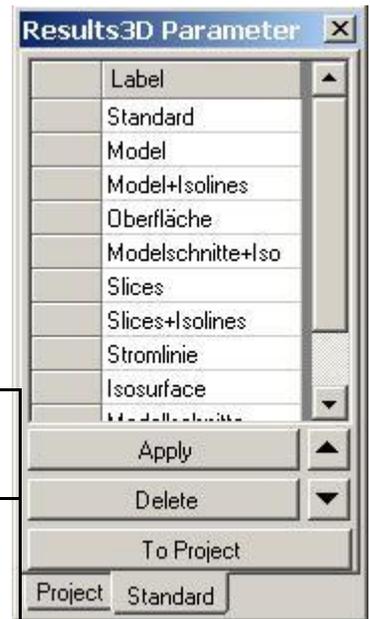
Results3D Parameter window

The window *Results3D Parameter* shows the list of collected 3D evaluation and visualization settings parameters. **Nearly all** parameters set within control panels are retained here as list entries. [Parameters specific to evaluation function](#) are not influenced from this window.

 Collecting new Results3D parameters is initiated from the [Results3D window](#) with the button [Collect Params \(in the control panel General\)](#) or from the [View menu](#).

The set of parameters shown as an item in the list can be easily restored and applied to the evaluation view.

The list of evaluation settings specific to the project is shown in the tab **Project**. The tab **Standard** shows the list of [application wide](#) evaluation settings.

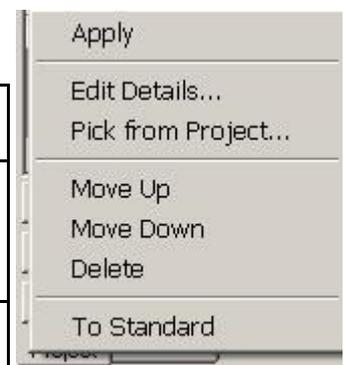


Label	An arbitrary, editable, label of respective item in the list of parameters (used for documentation and better, easier identification only)
Double-Click onto record-mark	Initiates the process of applying the combination of settings selected (marked line) to the 3D visualization. Remark.: Settings applied are taken from the current row – i.e. even if the click occurred within text of another line, the data applied is taken from the line marked with the record mark (an arrow symbol).
Apply	Initiates the process of applying the combination of settings selected (marked line) to the 3D visualization.
Delete	Deletes the currently selected (marked) item form the list.
Move Up ▲	The position of the item currently selected can be moved in the list.
Move Down ▼	The position of the item currently selected can be moved in the list.
To Standard	Settings combination associated with the selected entry within the project specific list will be copied (as additional entry) to the application wide list (Standard).
To Project	Settings combination associated with the selected entry within the application wide (Standard) list will be copied (as additional entry) to the project own list.
Project	Exposes the list of project specific evaluation settings
Standard	Exposes the list of application wide (standard) evaluation settings

Functions of the context menu

The display of the list has a context menu attached (exposed when right mouse button is pressed):

Apply	Initiates the process of applying the combination of settings selected (marked line) to the 3D visualization.
Edit Details...	Setting combination of the currently selected item can be edited in detail and adjusted according to specific needs. Remark: This is an advanced function. Naming chosen for distinctive parameters shall be self explanatory.
Pick from Project...	The list of project specific evaluation settings saved within some another project will be imported (appended) to the current project's list.
Move Up	The position of the item currently selected can be moved in the list.
Move Down	The position of the item currently selected can be moved in the list.
Delete	Deletes the currently selected (marked) item form the list.

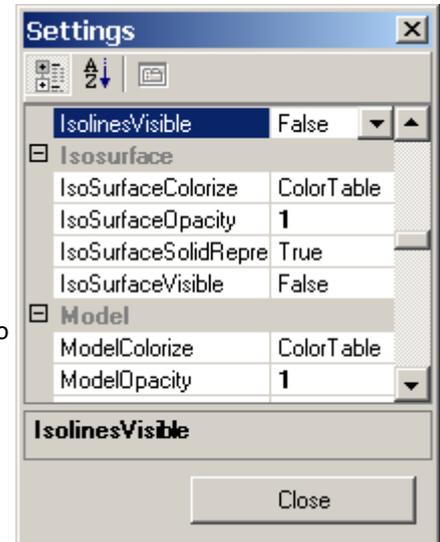


To Standard	Settings combination associated with the selected entry within the project specific list will be copied (as additional entry) to the application wide list (Standard).
To Project	Settings combination associated with the selected entry within the application wide (Standard) list will be copied (as additional entry) to the project own list.

Please note: Various settings listed in this window can be set within other control elements also.

The position of [slices X/Y/Z](#) can be set from [Probe points window](#) (see also [control panel Probe](#)).

The [camera position \(isometry\) of 3D view](#) can be set from [Isometries window](#) (see also [control panel 3D](#)).



Remark: To retain the list and the evaluation settings of the project between application executions you must [save the project!](#)

Remark: The application wide list of evaluation settings (Standard) will be [saved with application settings](#) (at application's exit).

See also: [Results3D window](#), [General \(control panel\)](#), [application settings file AnTherm.exe.Settings.xml](#), [Function specific evaluation parameters](#), [Settings - various \(dialog\)](#)

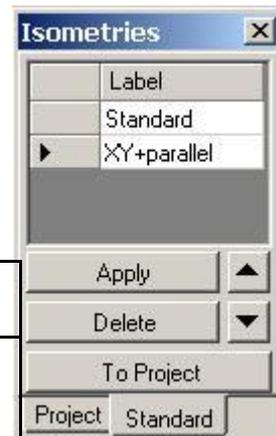
Isometries window

The window *Isometries* shows the list of collected 3D transformations of the 3D view.

Collecting new isometries is initiated from the [Results3D window](#) with the button [Collect Isometries \(in the control panel 3-D\)](#) or from the [View menu](#).

The isometry shown as an item in the list can be easily restored and applied to the 3D view.

The list of isometries specific to the project is shown in the tab **Project**. The tab **Standard** shows the list of [application wide](#) isometry settings.



Label	An arbitrary, editable, label of respective item in the list of isometries (used for documentation and better, easier identification only)
Double click onto record mark	Initiates repositioning the current view to the position (3D transformation) clicked within the list. Remark.: Transformation applied is taken from the current row – i.e. even if the click occurred within text of another line, the data applied is taken from the line marked with the record mark (an arrow symbol).
Apply	Initiates repositioning the current view to the position (3D transformation) currently marked within the list.
Delete	Deletes the currently selected (marked) item form the list.
Move Up	The position of the item currently selected can be moved in the list.
Move Down	The position of the item currently selected can be moved in the list.
To Standard	The isometry (3D transformation) associated with the selected entry within the project specific list will be copied (as additional entry) to the application wide list (Standard).
To Project	The isometry (3D transformation) associated with the selected entry within the application wide (Standard) list will be copied (as additional entry) to the project own list.
Projekt	Exposes the list of project specific isometries
Standard	Exposes the list of application wide (standard) isometries

Remark: Hold one of the modifier keys (Alt, Ctrl or Shift) depressed while pressing "Apply" or double-clicking the choice to view animated transformation of the position. The duration of the animation is set via the advanced application setting ["DesiredFlyToSeconds"](#).

Functions of the context menu

The display of the list has a context menu attached (exposed when right mouse button is pressed):

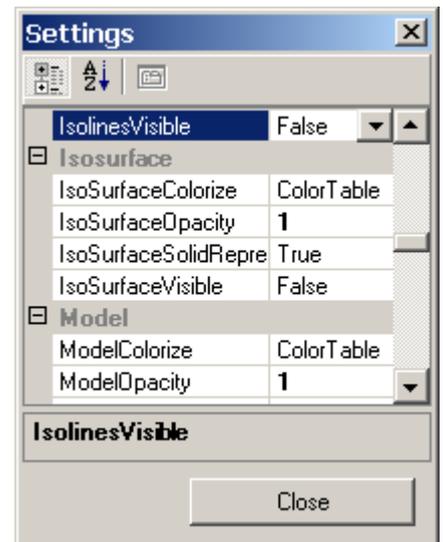
Apply	Initiates repositioning the current view to the position (3D transformation) currently marked within the list.
Edit Details...	Transformation data of the currently selected item can be edited in detail and adjusted according to specific needs. Remark: This is an advanced function. Naming chosen for distinctive parameters shall be self explanatory.
Pick from Project...	The list of project specific isometries saved within some another project will be imported (appended) to the current project's list. Remark: This function is available in the "Project tab" only.
Move Up	The position of the item currently selected can be moved in the list.
Move Down	The position of the item currently selected can be moved in the list.
Delete	Deletes the currently selected (marked) item form the list.
To Standard	The isometry (3D transformation) associated with the selected entry within the project specific list will be copied (as additional entry) to the application
To Project	The isometry (3D transformation) associated with the selected entry within the application wide (Standard) list will be copied (as additional entry) to the project own list



Please note: Various settings listed in this window can be set within other control elements also.

The position of [slices X/Y/Z](#) can be set from [Probe points window](#) (see also [control panel Probe](#)).

The [camera position \(isometry\) of 3D view](#) can be set from [Isometries window](#) (see also [control panel 3D](#)).



Remark: To retain the list and the evaluation settings of the project between application executions you must [save the project!](#)

Remark: The application wide list of evaluation settings (Standard) will be [saved with application settings](#) (at application's exit).

See also: [Results3D window, General \(control panel\)](#), [application settings file AnTherm.exe.Settings.xml](#), [Function specific evaluation parameters, Settings - various \(dialog\)](#)

Dialog windows

Within various dialog windows the user is requested for immediate input or response. Usually the execution of the application is interrupted until the user provides and confirm the requested input or cancels the operation explicitly.

Following dialog windows are shown within this application:

- Save/Open a file
- Print
- [Fine Grind parameter](#)
- [Settings](#)
- [Snap-To-Grid parameter](#)
- [Solver parameter](#)
- [Simulation folder cleaner](#)
- Colour
- [About info](#)
- [Revolve to 3D parameters](#)
- [Import of Image file \(Image Underlay\)](#)

See also: [Editing & Input windows](#), [Evaluation windows](#), [Main Menu](#), [The Main window](#)

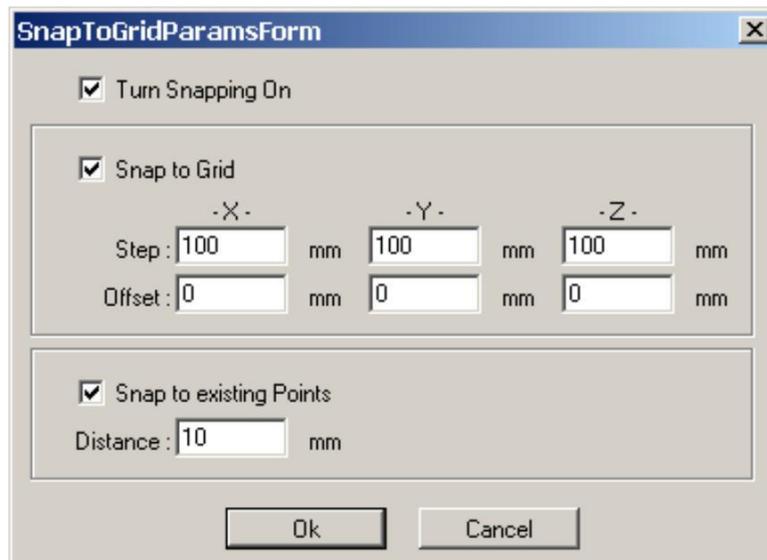
Snap-To-Grid parameter

The alignment grid, if active, restricts values of coordinates which can be reached by moving element boundaries (element's [editing markers](#)) with the mouse in [Elements 2D window](#).

An alignment grid can be set in such a way, that, for example, it only provides integral coordinates (each 1 mm) or, for example, at every 10 mm, etc.

Alternatively (or simultaneously) an alignment to already existing coordinates of the construction (within some snapping distance) can be requested also.

With alignment (snapping grid) turned off movements of [editing marker](#) will result in coordinate values calculated by the inverse transform of pixel coordinates at the mouse position. This transformation can result in numbers with large number of decimal places which, in most cases, will not correspond to desired element coordinates.



Turn Snapping On	Activates/deactivates the snap and align to grid function. It overrides the setting of specific alignment options below
Alignment Grid	
Snap to Grid	Activates/deactivates the snapping function on alignment grid. Positions of alignment grid lines can be calculated as follows: $Grid_coordinate_N = Offset + Step \cdot N$ where N is of integral type.
Step X/Y/Z	Defines the step of an alignment grid in millimetres. Must not be 0 ($> 1.0e-20$)
Offset X/Y/Z	Defines the starting point of an alignment grid in millimetres.
Point alignment	
Snap to existing points	Activates/deactivates the snapping function onto coordinates of points already existing in the construction.
Distance	Define the distance between the mouse position and neighbouring point coordinate below which snapping to the point coordinate will occur. Above this distance no snapping is applied and any arbitrary coordinate can be reached.

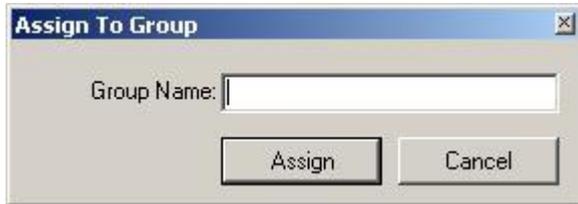
Alignment functions be activated and deactivated from the [context menu within Elements 2D window](#) also.

Settings of the alignment grid are saved together with [application settings](#).

See also: [Elements 2D window](#), [Application settings](#)

Assign To Group (Dialog)

The dialog window "Assign To Group" offers input of any (eventually new) group name to which all selected elements shall be assigned. You can reach this dialog from the [context menu of element editing](#) "Assign→To New Group...".



Remark: Group assignment is offered (for single elements) directly within [Element Editor](#) also.

Group Name	The name of a group to which all selected elements shall be assigned.
Zuweisen ENTER- Taste	The name entered will be assigned to all selected elements yet not in that group. The dialog window will close.
Abbrechen ESC-Taste	Closes the dialog window without executing any assignment.

Remark: By entering the name of an already existing group only elements yet not in that group will be assigned.

See also: [Element Editor](#), [Context menu of element editing](#)

Fine-Grid parameter (dialog)

The fine-grid subdivides the model into small cells which are then used to solve the thermal equations during simulation. Creation of the fine-grid is controlled via fine-grid parameters. Values offered by default have been preset in such manner, that for most usual cases none of them needs to be adjusted.

Settings of the fine-grid process directly influence the size of the equation system to be solved - the more detailed fine grid will create larger, eventually more complex, system of equations with higher number of cells to be balanced - that number is shown within the headings of [evaluation reports](#).

Important: Very fine and detailed gridding (discretization) does not automatically ensure higher precision of results. Too detailed discretization might lead to unnecessary and inefficiently long calculations or even to results with less precision.

The most important setting of [Start Step](#) shall typically correspond to the thickness of thinnest thermally significant structure of the construction. As such higher subdivision of metal slabs is not advisable.

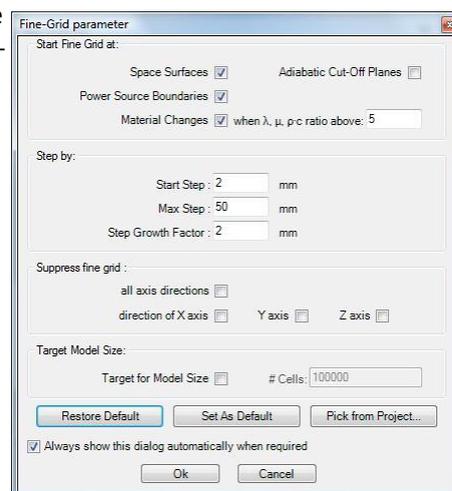
The [Maximum Step](#) shall be significantly raised when models of large dimensions of homogenous structures are considered.

The setting of [Target Model Size](#) is mostly **used while benchmarking the precision of results**, typically as last step during expertise process:

1. [coupling coefficients](#) and the number of cells of a calculation shall be identified within its [Coupling Coefficients report](#),
2. for the purpose of additional calculation of results the [Target Model Size](#) will be requested at the half or at the double of the cell count reported earlier and the new calculation is executed,
3. [Coupling coefficients](#) resulting from that second calculation (with half or double number of cells) are compared to the values received with the first calculation and respective precision criteria is validated then.

This final process of validation the precision of results conforms to the requirements set by the Standard EN ISO 10211.

Important: This validation is independent from the check of [results precision](#) provided within the respective [section of the Coupling Coefficients report](#) (the [relative Close-Up Error](#)).



Start Fine Grid at:	These are parameters defining starting criteria of the rasterizing process (creation of the fine grid)
Space Surfaces	Rasterizing will be initiated with its start step at each space surface and continue towards interior of the construction.
Adiabatic Cut-Off Planes	Rasterizing will be initiated with its start step at each adiabatic cut-off plane (adiabatic boundary) and continue towards interior of the construction. (in earlier application versions also called "Cut-Off/Wonder" foil)
Power Source Boundaries	Rasterizing will be initiated with its start step at each boundary line of any power source.
Material Changes	Rasterizing will be initiated with its start step at boundary lines of different materials. An additional start criteria is defined the quotient of conductivity values (lambda quotient).
when λ, μ, ρ-c ratio above	Rasterizing will start with its start step at boundary lines of different materials (material change) if the ratio of conductivity values (λ, μ, ρ-c values) of these materials is larger than the magnitude entered here. If the change of λ, μ and ρ-c is not significant (quotient is smaller the the value entered here) rasterizing will continue with its normal coarsening process. Remark: To force the rasterizing to start at each material change boundary set this parameter <=1 Remark: The ratio (quotient) of conductivities of two materials is always calculated as the division of the larger value by the smaller one, thus it always results in ratios values >= 1.

Step by:	
Start Step	Rasterizing will start with this distance when starting criteria is matched at specific position. Remark: This is not the smallest step within the resulting fine grid - smaller structures can result relative position of elements and element dimensions also!
Max Step	The process of coarsening the fine grid away from start positions does not go beyond this value. If applicable rasterizing will continue with this maximum value Remark: This value can be estimated automatically if target model size (number of fine grid cells) is chosen.
Step Growth Factor	At each further rasterizing step away from the position where the starting criteria has been matched the step of fine grid (distance) is enlarged by that factor. Remark: ≥ 1.0
Suppress fine grid in direction of X/Y/Z axis	
	Suppresses the process of fine grid generation in that specific axis direction. Resulting fine grid is then identical to minimal model grid. Remark: The application automatically detects homogenous dimension of the model (in that direction there will be no heat flow nor temperature gradients). If such direction is identified already within minimum grid, fine grid generation in this direction is automatically suppressed (regardless of the setting of the respective parameter here).
Target Model Size:	
Target for Model Size	When checked, one can request the target number of equations (cells). Rasterizer will then vary the maximum step to match the size at its best
# Cells	Rasterizer will vary the maximum step by an iterative search until it reaches the number fine grid cells within +/- 10% estimation range. Remark: If the target number cannot be reached sufficiently (i.e. maximum step could not be found satisfying other criteria), try to change the coarsening factor (step growth factor) . Remark: „Max Step“ will be varied between „Start Step“ and full model extent. If there is no solution sufficiently matching the targeted criteria the most coarse fine grid (in the case the target number was too small) or the finest fine grid (if the target was too large) will be created.
Restore Default	Reset all rasterizer settings to values provided in the application settings
Set As Default	Current project setting will replace values provided in the application settings .
Pick from Project...	Fine Grid settings will be loaded from another project.
Always show this dialog automatically when required	If checked the application will automatically reveal this dialog on each rasterizer run (in preparation of each calculation).

Fine Grid parameters are saved to the [project file](#).

Detailed description of gridding procedure

At first the application creates a "minimal grid" resulting from the overlapping of all elements. [Overlapping of elements](#) might result in removal of some elements if those are completely overlapped. The resulting model can be observed within the [Elements3D window](#). The [Modelling-Report](#) will list all spaces, surfaces, power sources and materials actually used within final model.

Start Fine Grid at:

Space Surfaces Adiabatic Cut-Off Planes

Power Source Boundaries

Material Changes when λ , μ , $\rho \cdot c$ ratio above:

Then, the application "visits" all boundaries of the minimal grid (which represents elements entered) and identifies if they are one of:

- Space surface (space boundary)
- change of material
- power source boundary
- adiabatic boundary

Step by:

Start Step : mm

Max Step : mm

Step Growth Factor : mm

Having fine-grid parameters set, the application will start adding additional grid planes in both directions away from these boundaries:

- initially with starting step
- every next one in the distance enlarged by the step growth factor compared to the previous
- until the maximum step size has been reached, which will be used to subdivide the rest of the model.

Suppress fine grid :

all axis directions

direction of X axis Y axis Z axis

This process is executed in all three Cartesian dimensions if not suppressed.

Target Model Size:

Target for Model Size # Cells:

Step by:

Start Step : mm

Max Step : mm

Step Growth Factor : mm

Solver parameter (dialog)

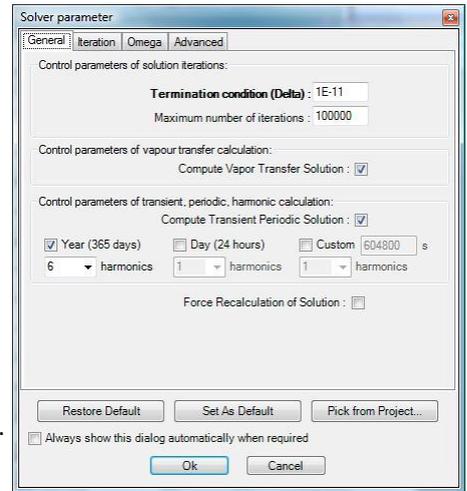
The Solver Parameter Dialog provides ability to adjust control parameters controlling directly the process of solving the equation system. Values offered by default have been preset in such manner, that for most usual cases none of them needs to be adjusted.

Provided the [VAPOUR-Option](#) and/or [HARMONIC](#) / [TRANSIENT](#)-Option are available the decision will be made here if [solving Vapour Transfer](#) and/or [Transient problems](#) shall be executed.

The most important setting is given by the [Termination Condition](#) which defines the targeted precision of the solution. This parameter is directly connected to the [resulting relative close-up error](#) describing the quality and precision of results. Eventually making the [Maximum Number of Iterations](#) larger might be needed to obtain the solution at requested precision in one solution run. Further parameters are used to optimize the efficiency (computational time) of the calculation process.

Within the [Advanced -tab](#) you can edit details of solver parameters or, for a very special case, adjust further advanced parameters.

Remark: The decision if the [solver shall run in parallel on multiple processors](#) has to be done within [application settings](#).

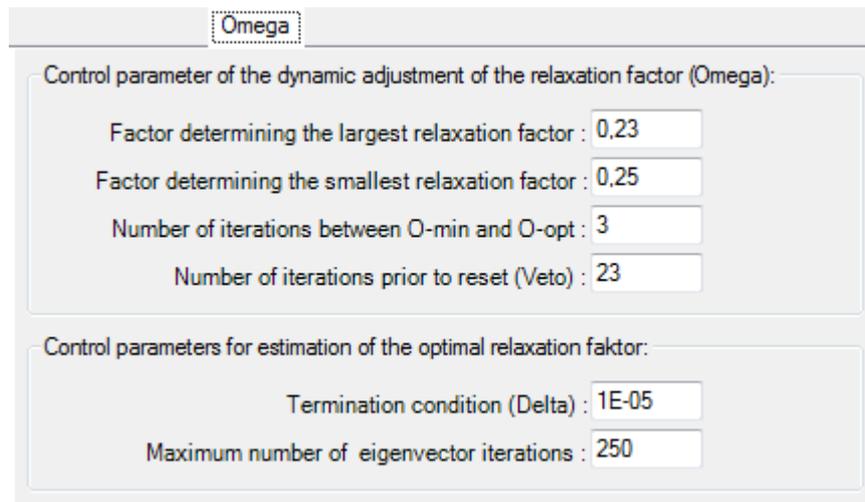


Control parameters of solution iterations (Iteration control)

<p>Termination condition (Delta)</p>	<p>If the largest change of temperature (normalized onto value interval 0.0 - 1.0) is continuously below the value entered here then the iterations are assumed as being precise.</p> <p>Reducing this value leads to more precise results at cost of longer computational time.</p> <p>Within the application settings: Delta Value range: $[5.0 \cdot 10^{-14}, 1.0]$</p>
<p>Maximum number of iterations</p>	<p>If the current number of iterations reaches the value entered here the solution process will be aborted without creating the solution.</p> <p>Remark: At each new run counting of iterations starts at 0.</p> <p>Since AnTherm Version 8 this value is fixed to 1 billion and cannot be changed by the user.</p>
<p>Minimum number of iterations below Delta</p>	<p>Defines the number of iterations at which the termination condition (delta) must be continuously satisfied prior to the assumption of precise solution and its final generation.</p> <p>Within the application settings: MinItNoBelowDelta Value range: $[1, 10000]$</p>
<p>Relaxation factor for initial</p>	<p>Defines the value of the relaxation factor ω which is used during initial "StartItNo" of iterations.</p>

iterations	Within the application settings : StartOmega Value range: [1.0 , 2.0]
Number of initial iterations	Defines the number of initial iterations with constant relaxation factor "StartOmega" (executed prior to further iterations with varying relaxation factor). Within the application settings : StartItNo Value range: [0 , 1000000]
Relaxation factor for final iterations	Defines the value of the relaxation factor ω which is used during final "FinalItNo" of iterations. Within the application settings : FinalOmega Value range: [1.0 , 2.0]
Number of final iterations	Defines the number of final iterations with constant relaxation factor "FinalOmega" (executed after "MinItNoBelowDelta" iterations with varying relaxation factor continuously satisfied the termination criteria - i.e. result variation was continuously below termination delta). This smoothes the final temperature distribution in the solution. Within the application settings : FinalItNo Value range: [0 , 1000000]
Number of iterations for tracing the Delta	Number of latest iteration steps to monitor the delta value changes. Within the application settings : DeltaTraceItNo Value range: [2 , 99], currently shall not be changed (read only) = 10.

Control parameter of the dynamic adjustment of the relaxation factor (ω) (Relaxtion factor control)



Factor determining the largest relaxation factor ω_{max}	Weighting factor used for calculation of largest relaxation factor ω_{max} which is larger than the optimal relaxation factor ω_{opt} . $\omega_{max} = \omega_{opt} - l * (\omega_{opt} - 1) * (\omega_{opt} - 2)$ Within the application settings : OmegaMaxFactor Value range: ≥ 0.0
Factor determining the smallest relaxation factor ω_{min}	Weighting factor used for calculation of smallest relaxation factor ω_{min} which is smaller than the optimal relaxation factor ω_{opt} . $\omega_{min} = \omega_{opt} + k * (\omega_{opt} - 1) * (\omega_{opt} - 2)$ Within the application settings : OmegaMinFaktor Value range: ≥ 0.0
Number of iterations between ω_{min} and ω_{opt}	Defines the speed at which the relaxation factor is increased between iterations. Within the application settings : OmegaMinOptItNo Value range: [2 , 1000]
Delta Trace ItNo	Number of latest iteration steps to monitor the delta value changes. Within the application settings : DeltaTraceItNo Value range: [2 , 99]
Number of iterations prior to	Even if the over-relaxation starts to be divergent (instead of converting to stable solution) the relaxation factor ω will be reset to its minimum value ω_{min} after at least that defined number of iterations between

reset (ω Veto)	consecutive resets. Within the application settings : OmegaVetoItNo Value range: [2 , 1000]
---	---

Control parameters for estimation of the optimal relaxation faktor (ω_{opt})	
Termination condition (Delta)	If the largest change to elements of the eigenvector of the equation matrix is continuously below this value the estimation of the optimal relaxation factor ω_{opt} is assumed to be precise and used for further calculation. Within the application settings : OmegaDelta Value range: [5.0*10 ⁻¹⁴ , 1.0]
Maximum number of eigenvector iterations	If the current number of iterations reaches the value entered here the estimation of the optimal relaxation factors ω_{opt} is terminated and the result estimated so far (even if not precise enough) is used in further calculation. Within the application settings : OmegaMaxNoOfIterations Wertebereich: [0 , 1000000]

Control parameters of vapour transfer calculation	
Compute Vapour Transfer Solution	If checked the application will create the solution of vapour transfer problem also (if applicable). Usually used to suppress vapour calculation even if all data required for such is available but that specific solution is not required. Within the application settings : "Compute vapour transfer solution" Remark: This setting requires a valid license feature DAMPF2DIM or DAMPF3DIM . 

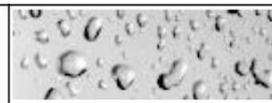
Control parameters of transient, periodic, harmonic calculation	
Compute Transient, Periodic Solution	If checked the application will create the solution of the transient, periodic, harmonic problem also (if applicable). Usually used to suppress transient calculation even if all data required for such is available but that specific solution is not required. Within the application settings : "Compute Transient Solution" Remark: This setting requires a valid license feature HARMONIC and/or TRANSIENT
Year (365 days) Day (24 hours) Custom (seconds)	Defines for which period length (year, day, custom) there will be harmonic computation executed. For the custom period enter the period length in seconds (1..99999999). Within the application settings : "PeriodYearActive", "PeriodDayActive", "PeriodCustomActive" Remark: This setting requires a valid license feature HARMONIC or TRANSIENT . Remark: Additional period lengths can be entered, if required, within advanced solver parameters (in seconds).
# harmonics Year/Day/Custom	Defines how many harmonics for the respective period length shall be calculated for the harmonic, transient solution. Year: max of 365 harmonics, Day: max of 24 harmonics, Custom: max of 1000 harmonics. With HARMONIC option alone (without TRANSIENT option) only the maximum of first 6 harmonics will be calculated. Within the application settings : "PeriodYearHarmonics", "PeriodDayHarmonics", "PeriodCustomHarmonics" Remark: This setting requires a valid license feature HARMONIC or TRANSIENT . Remark: Current implementation enforces same number of harmonics for every boundary conditions to be calculated regardless if that number of harmonics will be used during evaluation.
<div style="border: 1px solid #ccc; padding: 10px; background-color: #f9f9f9;"> <div style="display: flex; justify-content: space-around; margin-bottom: 10px;"> Restore Default Set As Default Pick from Project... </div> <p><input type="checkbox"/> Always show this dialog automatically when required</p> <div style="display: flex; justify-content: center; margin-top: 10px;"> Ok Cancel </div> </div>	
Force Recalculation of Solution	The existing solution (if available for loading) will be marked as "not solved" and calculation will be started. The calculation will restart with already available results. This setting will not be saved and is reset on each run. Remark: Usually the calculation step is skipped if valid results are already available. This setting provides

	the means to force calculation to start at any case to, for example, repeat the final "smoothing" iterations.
Restore Default	Resets all solver parameters to the values define within application settings
Set As Default	Current solver settings are set as default and stored as application settings
Pick from Project...	Solver parameters are loaded from another project file.
Always show this dialog automatically when required	Solver parameters are loaded from another project file.

Solver parameters are saved to the [project file](#).

Remark: Proper estimation of the optimal relaxation factor ω_{opt} becomes very important for large and possibly converging slowly calculations. It is advised to adjust parameters used for controlling the estimation of optimal omega in such a way, that better estimation ω_{opt} of is provided - reduce the termination condition of omega estimation (e.g. 10^{-7}) and enlarge the maximum number of iterations used in this process (e.g. 5000). The preset value of only 250 iterations provides very coarse estimation. Especially calculations of large cases with many spaces shall be run with well estimated omega because the process of solving by [over-relaxation](#) is executed repeatedly for each separate space! By using optimal relaxation factor the convergence behaviour of the solver will be sped up significantly!

VAPOUR-option: Analysis of multidimensional vapour diffusion is only possible with an active VAPOUR-Option of the program..



Running Solver on multiple CPUs

Running the [solver](#) on multiple CPUs enhances the calculation performance and thus (in most cases) reduces significantly the calculation time. [Multi CPU \(or Multicore\) behaviour of the solver is controlled by respective application settings](#) separately.

MULTICORE-option: Speeding up computationally intensive jobs by distributing them on multiple processors or processor cores for parallel execution is only possible with an active MULTICORE-Option of the program.



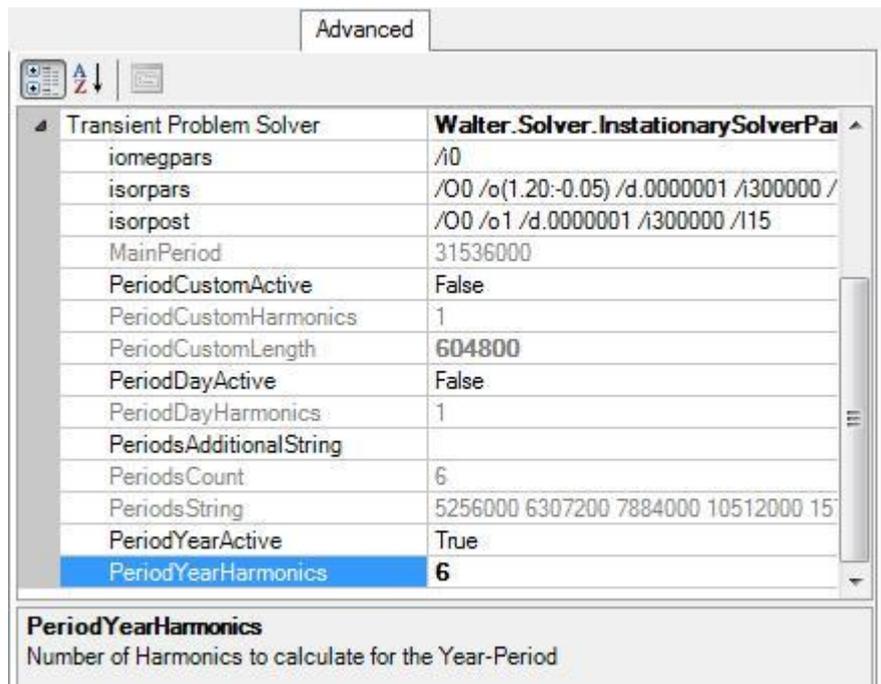
Advanced Solver parameters

Within the tab *Advanced* you can edit details of solver parameters.

Parameter settings composed of several parameters are shown pre-pended with a small plus sign. A click onto the plus sign will expand the detailed list of these parameters and offer them for editing.

The bottom part of the window shows explanations related to the parameter currently marked for editing. You can change the height of this description area by dragging the separator line shown above it.

Important: Transient (harmonic) solver parameters are grouped together within the section "Transient Problem Solver". Parameters offered by default shall not be changed. Ability to modify these settings is provided for testing and diagnostics purposes only.



Control parameter of the transient, harmonic Solver	
PeriodDayActive	Request harmonic calculation for the period of one day. Default: False
PeriodYearActive	Request harmonic calculation for the period of one year. Default: False
PeriodsAdditionalString	Request harmonic calculation for additional periods in seconds (separated by spaces). Default: ""
PeriodsCount	Displays count of requested periods.
<i>PeriodString</i>	Displays the list of requested period lengths.
<i>iomegpars</i>	Currently no meaning. Default: "/i0" Important: Do not change.
<i>isorpars</i>	Iteration control of the transient, harmonic solver. Default: "/O0 /o(1.20:-0.05) /d.0000001 /i300000 /i10" Important: Do not change.
<i>isorpost</i>	Iteration control for final iteration of the transient, harmonic solver. Default: "/O0 /o1 /d.0000001 /i300000 /i15" Important: Do not change.

Parameters displayed *slanted* will be passed verbose to the solver. Do not change.

Remarks:

- The highest number of periods the solver can process is limited to 2000.
- Period lengths (also of the higher harmonics) are kept as integers (seconds, 1..99999999).
- The "Periods" Array will contain more then only the fist 6 harmonics exclusively if TRANSIENT option is in place. Additional periods will be appended only if TRANSIENT is available.

See also: [Application settings](#), [Fine Grid parameter](#), [The over-relaxation](#), [Solver window](#)

Revolve to 3D Params - dialog

The conversion by revolution of a [2D project](#) to [3D model](#) is defined by the parameters provided in this dialogue window.

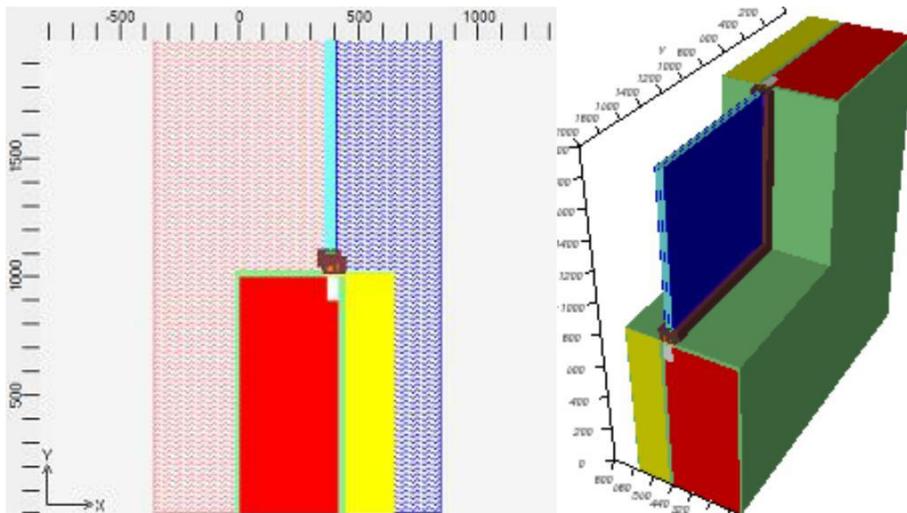
A 3D model is constructed by revolving the 2D object by 90° around vertical or horizontal rotation axis placed in the plane of the 2D model. The position of the axis relative to the 2D model defines the resulting 3D constructions. The option of clipping the 2D model to one side of the axis is also provided.

Important: Remember to [save the originating 2D model](#) prior to executing the revolution. The operation is not reversible and cannot be undone.

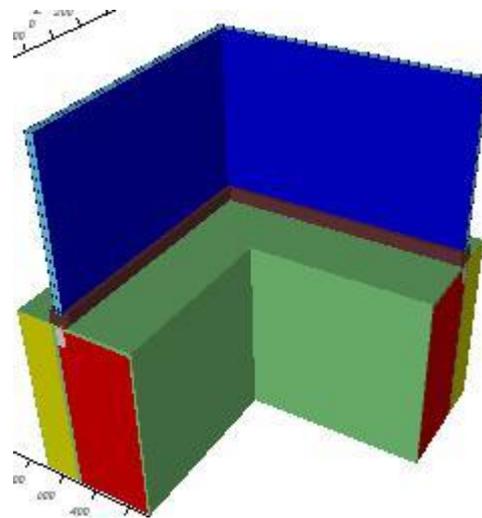
Revolution Axis Direction	Defines the orientation of the rotation axis the revolution shall be performed around. Choose either X (horizontal) or Y (vertical) axis direction.
Revolution Axis Position	Sets the position of the axis in absolute coordinates. Typically it will be some adiabatic boundary at some model extent or any arbitrary position for specialized purposes.
Revolved Axis Side	Elements of the 2D model revolved are clipped to either side of the axis. If only one side is chosen for revolution elements clipped for the other side are not revolved.
Z Offset	The resulting 3D model is constructed in such a way, that the Z position of the revolution axis will be kept equal with the axis position specified at first. Z Offset adds a post processing option to shift all resulting Z coordinates in the 3D model by the offset specified.
Revolve to 3D	Executes the revolution using the current set of parameters
Cancel	Aborts the operation

Examples

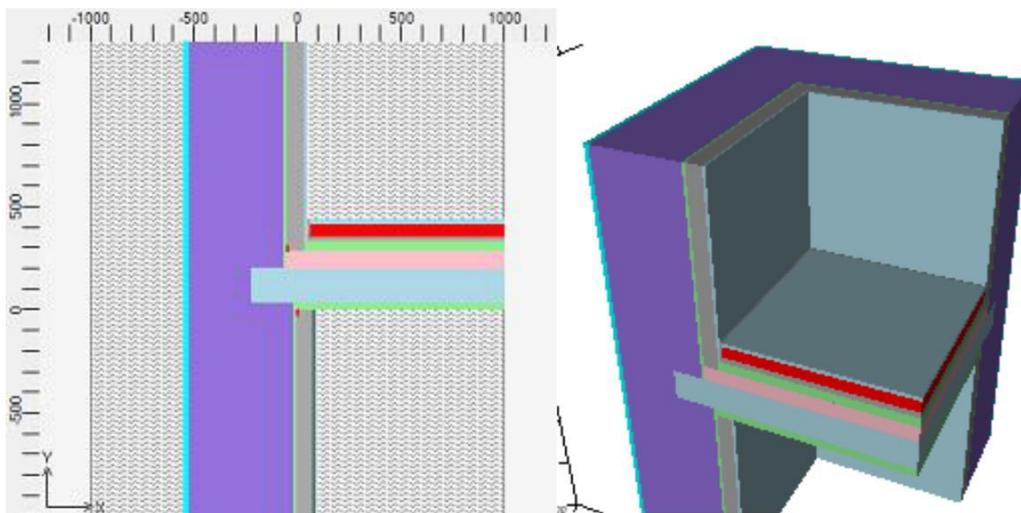
Example 1: Following 2D model is once converted by revolution around the X axis at Y=2000mm (which is coincident with the adiabatic boundary at the top of the 2D model)



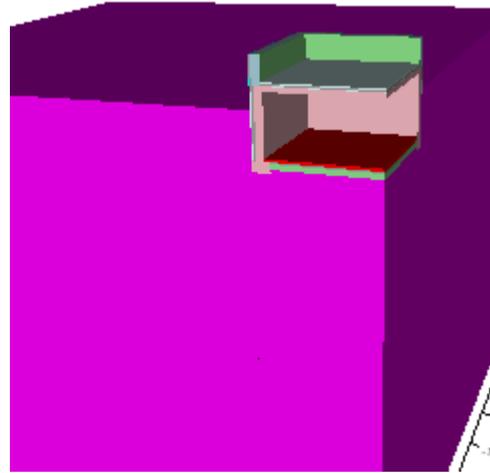
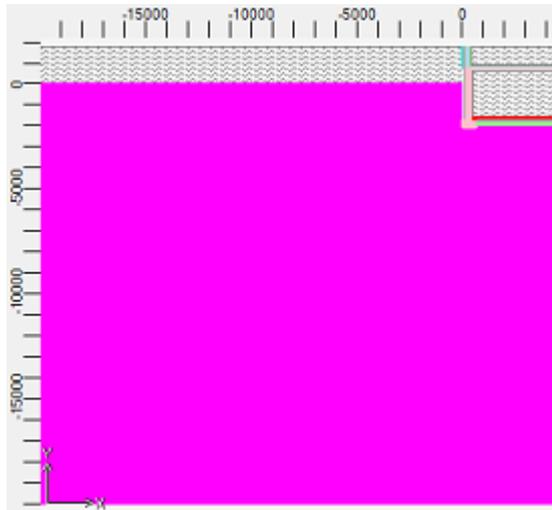
The model below results from the same 2D model revolved around the Y axis at X=-1000 mm.



Example 2: The 2D model of a wall & floor junction (shown below) is revolved around the Y axis at X=1000 mm producing a 3D corner component available for further detailed modelling and thermal assessment.



Example 3: Component in connection with ground was prepared in 2D at first and then revolved to 3D (rotated around Y axis at X=4475 mm).



See also: [Convert projects](#), [Project types](#)

About dialog window

The About dialog window displays current information related to the system environment, the application version, the license and copyright data. You reach the About window from the [main menu](#), *Help*→*About...* .

Remark: If the application executes within a [demonstrational mode](#) (if there is demonstrational license only available to the application), this window is exposed periodically in short time intervals. Purchasing a permanent or time limited license will turn this behaviour off.

The window is divided in three parts, each of which can be sized arbitrarily.

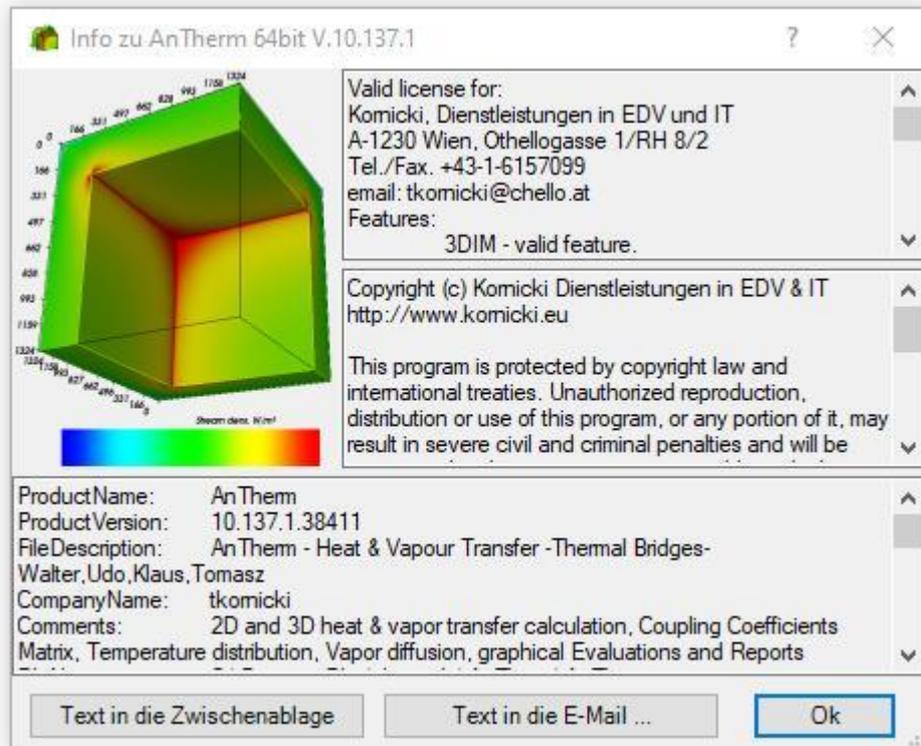
1. The first part, located at the top of the window, provides detailed information concerning the [application license](#) and validity of the license. This data is related to the [terms and conditions of use](#) of this application.

In the case the application shows information about missing license or tells, that the license is not valid you might be using an illegal version of the application.

Please immediately contact the vendor or copyright holder of the application in the case you see error messages in this part of the window.

Within this information block you will also find listing of all [hardware identifiers available for licensing](#) corresponding to the physical equipment installed in your computer (network adapters, processors, disks, license dongles, etc.).

2. The second part, located in the centre of the window shows valid vendor and copyright messages.
3. The third part, located at the bottom of the window, displays data about the version of the application and current operating environment. This data might be helpful during problem analysis or to clarify technical questions.



Text shown (or parts thereof) can be selected with the mouse and copied to the clipboard (via the context menu) for further processing.

Pressing the button **"Copy Text to Clipboard"** will copy text contents of all window parts onto the clipboard for further processing (e.g. to paste them into an e-Mail application).

The button **"Copy Text to E-Mail"** will start default e-Mail application setup on your computer (if possible) and pre-fill fields "To:" and "Subject:" with preset values furthermore inserting the e-Mail body with contents of all text modules from all parts of the about window.

The **departure of the e-Mail itself must still be confirmed by the user within the e-Mail application explicitly and intentionally** - AnTherm does not initiate e-Mail transfer by itself under no circumstances.

Remark: The addressee of the e-Mail preset in your version of the application might be antiquated or even not valid any more. Please verify the "To:" field prior to dispatching the e-Mail to make sure that the departure of the message will work.

Remark: Text contents will be determined within a background job. During that calculation period the copy buttons (clipboard, e-Mail) are shown disabled.

Remark: Contents of all parts might scroll automatically line by line at the interval of about 1 second. To stop the scrolling click with the mouse within the text part of interest or onto the scrollbar of that part.

See also: [System requirements](#), [License file AnTherm.HID](#), [License terms and conditions](#)

Simulation folder cleaner

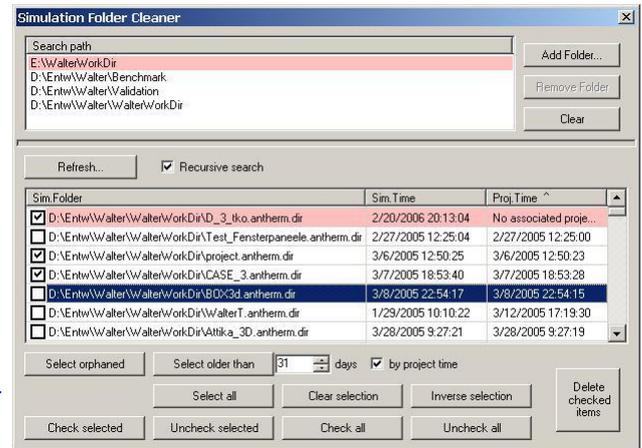
The simulation folder cleaner assists you by removal of [folders storing intermediate simulation results created by AnTherm](#).

During every calculation large amounts of tentative data is written to [simulation results folder](#) created for each [project](#) to speed up future repeated evaluations.

You shall remove these intermediary data from your disc regularly to save space. The data can be recreated in full from the data stored within respective project file at cost of some additional calculation time.

Using that cleanup tool consists of three steps:

1. Identify directories (potentially) containing [simulation results folders](#) to search for.
2. Select and mark for removal particular [simulation results folders](#).
3. Actually delete folders marked for removal.



This tool can be requested from the [menu Tools](#).

Remark: This tool is used to manage the [list of folders known to AnTherm as containers](#) of [simulation results folders](#) updated on each calculation run.

Important: During initial display of the window folders known to AnTherm are not searched automatically - the *Refresh* button will blink in short intervals. Please press the *Refresh* button to initiate the search - the application will block and show wait cursor during this period. Actually, the search process cannot be interrupted indeed.

Remark: If the number of days passed since the last invocation of this tool is above the number set by the [application setting "Simulation Folder Cleanup \(Reminder\)"](#), then on the next application start-up a reminder message will be shown.

Identifying folders potentially containing simulation results folders (search folder names)

Top part of the window shows the list of directories to be later searched for [simulation results folders](#). Initially the list shows [folders known to AnTherm as containers](#) of such [simulation results folders](#) which is stored within the [application settings file](#) and updated on each calculation run.

Search folders	Displays the list of directories expected to contain simulation results folders . Click onto an item to select it for eventual removal. Multiple selections are supported (use Ctrl- or Shift- key combination). Remark: Inaccessible directories are highlighted in red - you shall eventually consider removing them manually from the list.
Add...	Will expose the directory browser helping you in selecting an existing folder to add to the search list
Remove	Will remove selected items from the search list. Remark: Removing an item from the list will not delete that directory.
Clear	Will remove all items from the search list. Remark: Removing an item from the list will not delete that directory.

Important: The search list resembles the [application setting "Known simulation folder containers"](#) which is currently edited in this tool.

Searching for simulation results folders

The central part of the window displays the list of folders found during the search and provides the means for marking them for removal also.

On every change to the [search list](#) the application will search for [simulation results folders](#) in directories specified (actually only folders added to the search list will be searched to improve performance).

Remark: Searching directories might last for several minutes - the application will block and show wait cursor during this period.

Remark: The search cannot be interrupted.

Refresh...	<p>Will execute full search within all search paths.</p> <p><u>Remark:</u> On each change to the search list the search is executed automatically.</p>
Recursive search	<p>If marked, the search will include folders specified in the search list and all subdirectories of these folders.</p> <p><u>Remark:</u> If the search list contains paths which are contained within the other some items found will be shown duplicate - this has no general impact on function of the tool currently.</p>
List of simulation folders	<p>Displays the list of simulation results folders found during the search in search folders together with its modification time (time of last calculation) and the time the project file was last written (also time of latest evaluation).</p> <p>Click onto an item to select it. Multiple selections are supported (use Ctrl- or Shift- key combination).</p> <p>Click on column title to update the sorting direction (initially items are shown not sorted).</p> <p>Check the Checkmark left to the folder name to mark it for removal.</p> <p><u>Remark:</u> Orphaned simulation results folders (i.e. those lacking their associated project file) are highlighted in red and are automatically marked for removal.</p>

Selecting and marking simulation results folders for removal

The bottom part of the window provides the means for selecting and marking items for removal and deleting them finally.

Selection buttons	
<u>Remark:</u> Selection does not mark items for deletion, its supports it only.	
Select orphaned	Will select orphaned simulation results folders only (i.e. those lacking their associated project file).
Select older than days	Will select simulation results folders by either their simulation time or the project time. Number of days. Only folders older than number of days will be selected.
by project time	Selection of old folders is done by comparing the time the project file was last written (also time of latest evaluation) by default. If not marked (unchecked) the last simulation time will be used.
Select all	Will select all items
Clear selection	Will remove the selection from all items.
Inverse selection	Will reverse the selection on every item (those selected will become unselected and vice-versa).
Marking for removal buttons	
<u>Remark:</u> Marking items for deletion does not remove items yet.	

Check selected	Will mark selected items for removal.
Uncheck selected	Will remove the deletion mark from selected items.
Check all	Will mark all items for removal.
Uncheck all	Will remove the deletion mark from all items.
Final Removal	
Delete checked items...	<p>Will start the process of irrevocable removal of items checked. You will be asked to confirm the request.</p> <p>If errors occurred during the actual removal process a dialog message will show the error report after the process has been finished for all items marked for removal. Items deleted successfully will be removed from the list also. Items in error will stay marked in the list.</p> <p><u>Remark:</u> Actual removal of simulation results folders might last for several minutes - the application will block and show wait cursor during this period.</p> <p>Warning: The removal process cannot be interrupted nor it is reversible.</p> <p><u>Remark:</u> Removing simulation results folders does not influence the search list. Eventually you shall remove specific directory names from the search list manually if you do not expect them to contain simulation folders in the future.</p>

KnownSimulationFolderContainters (application setting)

On each [evaluation](#) execution (and any potential calculation) AnTherm will add the parent directory of the [project file](#) (and thus the parent directory of the [simulation results folder](#) too) to the application setting "KnownSimulationFolderContainters". This provides the notion of folders being potential containers of [simulation results folders](#) and is retained between program executions (stored within the [application settings file](#)).

The setting "KnownSimulationFolderContainters" is not managed within [application setting window](#) but within this "Solution Folder Cleaner" here. Folder names are shown as "[Search folders](#)" within this tool.

Important: When the window of this tool is closed the application setting will receive contents of the list as entered and shown. This tool provides the only means to remove directories from that list (or add items to it) manually.

Heuristics used to identify simulation results folder

The application uses following heuristics to identify a folder as an AnTherm's [simulation results folder](#):

- The name of the folder ends with „*dir*“.
- Following files exist within that folder at least:
C.dat, dim.dat, I_.dat, J.dat, K_.dat, L.dat, L_.tmp, LS.tmp, N.dat, ROOMS.dat, S.dat, S_.dat, xyz.dat

See also: [Working with files](#), [Settings file AnTherm.exe.Settings.xml](#)

Please Wait window

This window is shown during long lasting jobs.



The number in braces shows the count of background jobs which have to be waited for. At the bottom of the window short information about executed jobs might display optionally.

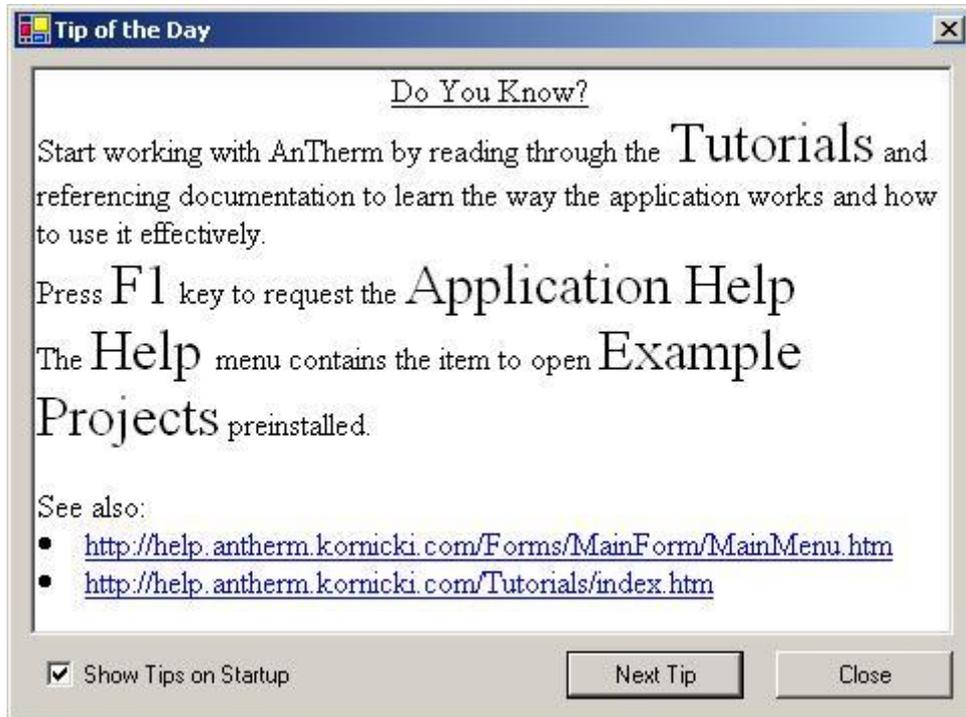
While this window is shown **no further interaction with the program is possible by intent**. Please wait...

Note: Under certain circumstances the application will start several evaluations executing simultaneously and asynchronously. When run on weaker computer systems this might lead to over utilisation of resources or even program lock up. To turn off such asynchronous execution of evaluations and let the program execute them synchronously, one after another, please change the [application setting "Asynchronous Evaluation Builds"](#).

See also: [Results 3D window](#), [Results report](#), [Coupling Coefficients report](#)

Tip of the day (Dialog)

The Tip of the Day displays some useful information within a dialog window. This hints are mostly helpful to new users of AnTherm, but even experienced users shall revisit this window from time to time to receive new short informations.



The dialog window is shown automatically on each application start (this can be turned off) or from the [Help Menu](#) "Tip Of The Day...".

Show Tips on Startup	Having this checkmark set the dialog window will show at each application start. Remark: This switch can be turned off only after all tips have been shown several times. This shall emphasize the importance of those tips for new users. (All tips have been watched at least 2 times for at least 10 seconds in average; or the user clicks 10 times through all tips. Data is stored within ApplicationSettings/UsageStatistics).
Next Tip	Show the next tip.
ENTER-key	Remark: After showing all text components installed with the program first tip will be shown again.
Schließen ESC-Taste	Closes the dialog window.

Remark: Additional information linked from the displayed content are often located on the internet. To display these additional content internet connection is needed.

See also: [Help Menu, Tutorials and Cookbooks](#)

Settings - various (dialog)

Detailed editing of parameter, settings and options for which there is no specific data entry window will be done by this window.

- detailed editing of [Results3D Parameters](#) of the graphical evaluation,
- detailed editing of [Isometries window](#) of the graphical evaluation,.

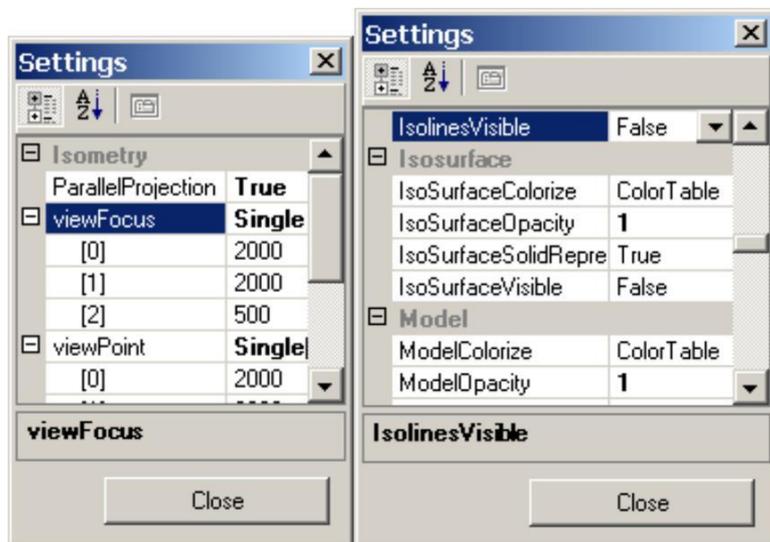
Remark: [The Application Settings](#) are separately available within the [Application Settings Dialog](#).

Parameters which are composed of a set of detailed settings are shown with a plus symbol to the left of the parameter name. A click onto the plus symbol will expand the list of detailed setting offering them for editing.

The bottom part of the window displays the description of the parameter currently selected. The height of this part of the window can be adjusted by dragging the splitter bar.

Settings shown and adjusted in this window are applied immediately to the application. Changes may also propagate to further settings. Changes to some parameters require an application restart (e.g. User interface language) - if so it is emphasized in the description specific parameter.

Hint: To restore the value of any respective parameter to its default use its context menu (right mouse button) and select "Reset".



See also: [Results3D Parameter window](#), [Isometries window](#), [Application Settings \(dialog\)](#)

Leitwert Lambda Equivalent Calculator (Lambda-EQ Tool)

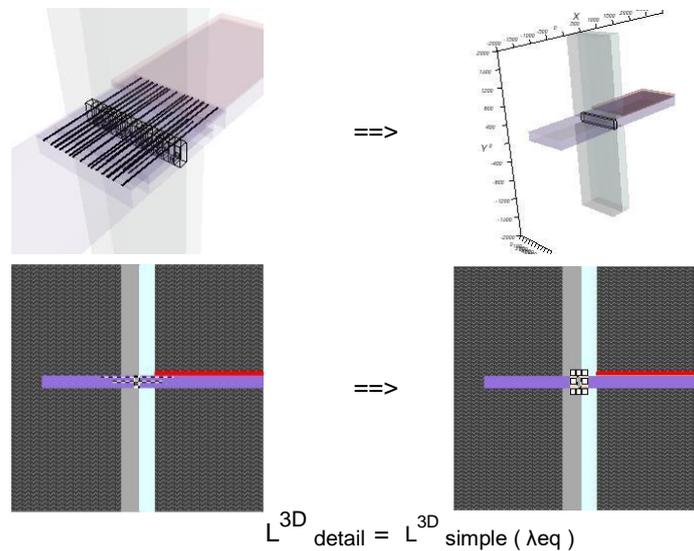
Objective

The Lambda Equivalent Calculator (Lambda-EQ tool) is used to calculate the thermal conductivity of a component consisting of a building material with a simplified replacement model of thermal separation (equivalent Lambda of the simplified model), in such a way that the thermal conductance of the simplified model must be equal to the conductance of the detail model.

If necessary, the detail model is available as a 3D construction, the simplified model as 2D only. In such a case $L^{3D}/dZ = L^{2D}$ must be used.

By variation of the building material property, the lambda-eq of a building material is to be calculated iteratively for an inputable or calculated thermal coupling coefficient (target value) in the two-room case.

Detail model ("Isokorb" K50CV35h200) Equivalent model (Insulating block)



The target conductivity value can be set as follows:

- Calculation from the current project (ProjectDetailName empty)
- Calculation from the specified project ProjectDetailName
- Numeric input (ProjectDetailName empty)

The thermal conductivity SimplifiedLambda of a building material SimplifiedMaterialNameToVaryLambda of a simplified model is to be calculated.

The resulting conductance SimplifiedLeitwert of the simplified model shall differ only slightly from the target conductance. The specification is made with the relative deviation SimplifiedConductanceIterationMaxRelativeDelta related to the magnitude of the conductance.

The simplified model can be specified as follows:

- Construction from the current project (ProjectDetailName empty, ProjectSimplifiedName empty) in which elements from the DetailGroupToRemove group are removed (the elements SimplifiedMaterialNameToVaryLambda are retained, however)
- Construction from the specified project ProjectSimplifiedName in which elements are removed from the DetailGroupToRemove group (the SimplifiedMaterialNameToVaryLambda elements are retained)

The iterative process ends when:

- The desired accuracy (precision) SimplifiedLeitwertIterationMaxRelativeDelta is reached (success)
- The specified maximum number of iterations SimplifiedLeitwertIterationsMaxCount is reached (no result)

Leitwert Lambda Eq = 0,1669391651153564453125 W/mK

a) Target Data:	
01: ProjectDetailName	...\detail.anthem
02: DetailLeitwert	0,761280195176535
03: ProjectSimplifiedName	...\simplified.anthem
04: DetailGroupToRemove	K50CV35h200
05: SimplifiedMaterialNameToVaryLambda	Dämmkörper
b) Result Data:	
01: SimplifiedLambda	0,1669391651153564453125
02: SimplifiedLeitwert	0,761312085523077
03: LeitwertAbsoluteDelta	0,000031890346542
04: LeitwertRelativeDelta	0,0000418904192491240033347457
c) Auxiliary Data:	
01: SimplifiedLambdaRangeMin	0,001
02: SimplifiedLambdaRangeMax	1000
03: SimplifiedLeitwertIterationsMaxCount	50
04: SimplifiedLeitwertIterationsCount	21
05: SimplifiedLeitwertIterationMaxRelativeDelta	0,0001

01: SimplifiedLambda
 (optional) Die Wärmeleitfähigkeit die dem Baustoff SimplifiedMaterialNameToVaryLambda des vereinfachten Modells zugewiesen wird. Im Zuge der Berechnung wird der Wert der Wärmele...

Reset Undo Cancel Start Done (Save)

When calling the Lambda-EQ tool from the menu, the current project must first be saved.

When exiting the tool, the project is restored (read in from the file). The settings and results made in the tool are automatically saved in the project (automatic saving again).

Reset	Resetting the control parameters to their default values
Undo	Resetting the control parameters to the values when the dialog is opened
Cancel	Abort the function
Start	Starting the calculation
Done/Save	Save settings and results in the current project

The Settings

Target Data:	
ProjectDetailName	(optional) Name of the project of the detail model for the calculation of the Conductivity DetailLeitwert. If empty, then the project currently being processed is used. Default: empty
DetailLeitwert	(optional) The thermal coupling coefficient of a detail model (and the target variable which is to be approximated by the conductance SimplifiedLeitwert of the simplified model If empty or 0, it is used for the detail model (ProjectDetailName or current project). If not empty, it is calculated with the conductance of the detail model (ProjectDetailName or current project). Default: 0
ProjectSimplifiedName	(optional) Name of the project of the comparison model (simplified model) for which the variation of the thermal conductivity SimplifiedLambda of a building material SimplifiedMaterialNameToVaryLambda results in one (approximately) identical thermal conductance SimplifiedLeitwert like the DetailLeitwert. If empty, the simplified model is created by removing the elements belonging to the group DetailGroupToRemove (but the elements SimplifiedMaterialNameToVaryLambda are not maintained) Default: empty
DetailGroupToRemove	(optional) The elements belonging to the group DetailGroupToRemove are to be removed from the simplified model (the Elements SimplifiedMaterialNameToVaryLambda are however maintained) Default: empty
SimplifiedMaterialNameToVaryLambda	(Mandatory field) Name of the building material from which the value of the Thermal conductivity SimplifiedLambda is to be adjusted (varied) until the thermal conductance SimplifiedConductance of the simplified model matches the conductance DetailLeitwert of the detail model except for a relative deviation SimplifiedLeitwertIterationMaxRelativeDelta. Default: empty
Results Data:	
SimplifiedLambda	(optional) The thermal conductivity of the building material SimplifiedMaterialNameToVaryLambda of the simplified model is assigned. In the course of the calculation, the value of the thermal conductivity SimplifiedLambda is adapted (varies) until the thermal conductance SimplifiedLeitwert of the simplified model matches the conductance DetailLeitwert of the detail model except for a relative deviation SimplifiedLeitwertIterationMaxRelativeDelta. If empty or 0 it is calculated. If not empty, it is inserted and checks whether it matches the desired accuracy criteria, and if not, recalculated. 0 as result: the calculation failed.
SimplifiedLeitwert	The thermal conductance of the simplified model ... 0 as result: the calculation failed.
SimplifiedLeitwertAbsoluteDelta	Amount of the difference between thermal conductance DetailLeitwert of the detail model and the SimplifiedLeitwert of the simplified model. 0 as result: the calculation failed.
SimplifiedLeitwertRelativeDelta	Conductance deviation related to the amount of thermal conductance DetailLeitwert. In the course of the calculation this is compared with the accuracy criterion SimplifiedLeitwertIterationMaxRelativeDelta. 0 as result: the calculation failed.
Auxiliary Data:	
SimplifiedLambdaRangeMin	In the course of the calculation, the value of the thermal conductivity SimplifiedLambda varies in the interval SimplifiedLambdaRangeMin to SimplifiedLambdaRangeMax. Default: 0.001 W/mK Should be < SimplifiedLambdaRangeMax

SimplifiedLambdaRangeMax	In the course of the calculation, the value of the thermal conductivity SimplifiedLambda varies in the interval SimplifiedLambdaRangeMin to SimplifiedLambdaRangeMax. Default: 1000 W/mK Should be > SimplifiedLambdaRangeMin
SimplifiedLeitwertIterationsMaxCount	The maximum number of iteration steps SimplifiedLeitwertIterationsCount in the course of the SimplifiedLambda search. Default: 50
SimplifiedLeitwertIterationsCount	The current number of iteration steps SimplifiedLeitwertIterationsCount in the course of the SimplifiedLambda search.
SimplifiedLeitwertIterationMaxRelativeDelta	The accuracy criterion and termination condition. The search for SimplifiedLambda is considered successful if the relative difference between the conductance DetailLeitwert of the detail model and the SimplifiedLeitwert of the simplified model (which is calculated from the thermal conductivity SimplifiedLambda) is below this limit. Default: 0.0001 (also corresponds to ISO 10211)

Further control elements

Following control elements or settings are common to several parts of the application and collectively listed:

- [3D Navigation within 3D windows](#)
- [Background colour \(setting\)](#)
- [More Light \(setting\)](#)
- [Solid or Wireframe \(setting\)](#)
- [Copy image to clipboard \(command\)](#)

See also: [Results 3D window](#), [Elements 3D window](#)

Toolbar of a Report window

Report windows expose common toolbar containing following symbols:



- [Export report](#)
- [Print report](#)
- Go to first page
- Go one page back
- Go one page forward
- Go to last page
- Go to page number entered
- Search
- Zoom (scale) the view

Exporting the report

Reports can be exported (saved) into following document formats:

- Adobe Acrobat PDF (.pdf)
- Character Separated Values (.csv)
- Microsoft Excel (97-2003) (.xls)
- Microsoft Excel (97-2003) Data-Only (.xls)
- Microsoft Excel Workbook Data-Only (.xlsx)
- Microsoft Word (97-2003) (.doc)
- Microsoft Word (97-2003) - Editable (.rtf)
- Rich Text Format (RTF) (.rtf)
- XML (.xml)

By pressing the symbol *Export*, (within the *Toolbar*), you can convert and save the report currently displayed into one of supported formats.

While the report window is shown you can use the [main menu's](#) submenu [File→Export→Report...](#) to initiate export of report data.

Printing the report

The printing function is available via the button *Print*, within the *Toolbar*.

While the report window is shown you can use the [main menu's](#) submenu [File→Print...](#) to initiate printing of current report.

See also: [Input report](#), [Results report](#), [Coupling Coefficients report](#), [Modelling report](#), [Evaluation windows](#), [Evaluations and Results](#)

3D Navigation within 3D windows

The imagery displayed in three dimensional rendering views can be controlled in various ways:

- With the [mouse](#)
- By [keyboard](#) shortcuts
- With [3D control panel](#) (within [Results3D window](#))

In addition immersive [stereo display rendering](#) in various technologies can be requested.

3D Navigation with the mouse

Graphics shown within 3D windows ([Elements 3D](#), [Results 3D](#)) can be manoeuvred in virtual 3D space (rotation, translation or scaling) with the mouse easily. The [level of display detail will be reduced](#) during the motion if required.

Remark: When the mouse pointer is located above some special objects within the view these objects can be specifically controlled directly in different manner then described here (e.g. [position of the movable colour bar](#), [moveable probe point X,Y,Z](#), [moveable orientation marker](#)).

Place the mouse pointer over graphics view of the window:

<p>Left mouse key, drag: Press and hold the <i>left mouse button</i>, move the mouse pointer.</p>	<p>A dragging action (holding down the left mouse button, then moving the mouse) rotates the view in 3D space. The view behaves as if it were attached to a virtual trackball whose centre is at the centre of the view window. The mouse cursor is attached to the part of the trackball that protrudes above the surface of the window. Similarly the view follows a virtual joystick held with the mouse while joystick behaviour is activated instead. You can switch between trackball- and joystick behaviour with keyboard controls.</p>
<p>Ctrl + mouse wheel turn Hold the <i>Ctrl key</i> and turn the mouse wheel or: Ctrl + left mouse button, drag: Hold the <i>Ctrl key</i> and press and hold the <i>left mouse button</i>, move the mouse pointer.</p>	<p>The 3D view is rotated around the axis in the geometrical centre of the view and perpendicular to the screen surface (rotation around the block axis, also called "Rolling"). You will need this function to flip the view "upside down" for example.</p>
<p>Shift + left mouse button, drag: Hold the <i>Shift key</i> and press</p>	<p>The view is translated in the plane of the window following the mouse pointer (translation, scrolling, panning in the view plane).</p>

and hold the <i>left mouse button</i> , move the mouse pointer.	
<p>Mouse wheel turn or: Shift + Ctrl + left mouse button, drag: Hold the <i>Shift</i> and the <i>Ctrl</i> keys and press and hold the <i>left mouse button</i>, move the mouse pointer.</p> <p>or: Right mouse button, drag: Press and hold the <i>right mouse button</i>, move the mouse pointer.</p>	<p>The view is scaled (zoom-in and zoom-out).</p> <p>Scaling operation is executed relative to the geometrical centre of the 3d view in virtual space</p>
 Press the Fit button	The position and scaling of the view will be adjusted in such a way, that the whole building component fits into the view.
 Press the Reset button	<p>Resets the camera (viewer) position to its default state:</p> <ul style="list-style-type: none"> • X/Y plane parallel to the view (window) plane. • Z axis points towards the viewer (perpendicular to window plane). • Y axis is oriented vertical and upwards. • X axis points to the right. <p>Scaling of the view is adjusted accordingly to fit the whole component into the view.</p>

3D Navigation with the keyboard

Graphics shown within 3D windows ([Elements 3D](#), [Results 3D](#)) can be manoeuvred in virtual 3D space (rotation, translation or scaling) with the keyboard. Further settings of the displayed graphics can also be controlled and adjusted with the keyboard.

Click once with the mouse onto the graphics part of the window to move the keyboard focus to it.	
Keyboard key	Function
T	Trackball Interactor of the mouse - dragging the mouse controls the moves of the 3d view
J	Joystick Interactor of the mouse - the position of the mouse in the view controls the direction and the speed of continuous movements
P	Perspective On/Off
L	More Light On/Off
F	Fit (also "NumPad 5")
+ (Plus)	Scale (Zoom) +10% (also "Numpad +")
- (Minus)	Scale (Un-Zoom) -10% (also "Numpad -")
Ins	Side view from the left
Del	Side view from the right
Home	Side view from the front
End	Side view from the back
Page-Up	Side view from the top (also "Enter")
Page-Down	Side view from the bottom
Enter	Side view from the top (also "Page-Up")
	Remark: Hold one of the modifier keys (Alt, Ctrl or Shift) depressed while pressing the button to view animated transformation of the position. The duration of the animation is set via the advanced application setting " DesiredFlyToSeconds ".

Numpad 5	Fit (also "F")
Numpad +	Scale (Zoom) +10% (also +, Plus)
Numpad -	Scale (Un-Zoom) -10% (also -, Minus)
Numpad 6	Rotate to the right around the vertical axis, Azimuth -10° (also Right-Arrow)
Numpad 4	Rotate to the left around the vertical axis, Azimuth +10° (also Left-Arrow)
Numpad 8	Rotate upwards around the horizontal axis, Elevation -10° (also Up-Arrow)
Numpad 2	Rotate downwards around the horizontal axis, Elevation +10° (also Down-Arrow)
Numpad 9	Rotate to the right/upwards around a diagonal axis, Azimuth/Elevation, 7,07°/7,07°
Numpad 7	Rotate to the left/upwards around a diagonal axis, Azimuth/Elevation, 7,07°/7,07°
Numpad 1	Rotate to the left/downwards around a diagonal axis, Azimuth/Elevation, 7,07°/7,07°
Numpad 3	Rotate to the right/downwards around a diagonal axis, Azimuth/Elevation, 7,07°/7,07°
Ctrl + Numpad 6	Roll to the right around the view (block) axis, -10° (also Ctrl + Right-Arrow)
Ctrl + Numpad 4	Roll to the left around the view (block) axis, +10° (also Ctrl + Left-Arrow)
Ctrl + Numpad 8	Roll to the right around the view (block) axis, -10° (also Ctrl + Up-Arrow)
Ctrl + Numpad 2	Roll to the left around the view (block) axis, +10° (also Ctrl + Down-Arrow)
Shift + Numpad 6	Translate (scroll, pan) to the right, -10 pixels (also Shift + Right-Arrow)
Shift + Numpad 4	Translate (scroll, pan) to the left, +10 pixels (also Shift + Left-Arrow)
Shift + Numpad 8	Translate (scroll, pan) upwards, -10 pixels (also Shift + Up-Arrow)
Shift + Numpad 2	Translate (scroll, pan) down, +10 pixels (also Shift + Down-Arrow)
Shift + Numpad 9	Translate (scroll, pan) diagonal to the right and upwards by 10 pixels
Shift + Numpad 7	Translate (scroll, pan) diagonal to the left and upwards by 10 pixels
Shift + Numpad 1	Translate (scroll, pan) diagonal to the left and down by 10 pixels
Shift + Numpad 3	Translate (scroll, pan) diagonal to the right and down by 10 pixels
Right-Arrow	Rotate to the right around the vertical axis, Azimuth -10° (also Numpad-6)
Left-Arrow	Rotate to the left around the vertical axis, Azimuth +10° (also Numpad-4)
Up-Arrow	Rotate upwards around the horizontal axis, Azimuth -10° (also Numpad-8)
Down-Arrow	Rotate downwards around the horizontal axis, Azimuth +10° (also Numpad-2)
Ctrl + Right-Arrow	Roll to the right around the view (block) axis, -10° (also Ctrl + Numpad-6)
Ctrl + Left-Arrow	Roll to the left around the view (block) axis, +10° (also Ctrl + Numpad-4)
Ctrl + Up-Arrow	Roll to the right around the view (block) axis, -10° (also Ctrl + Numpad-8)
Ctrl + Down-Arrow	Roll to the left around the view (block) axis, +10° (also Ctrl + Numpad-2)
Shift + Right-Arrow	Translate (scroll, pan) to the right, -10 pixels (also Shift + Numpad-6)
Shift + Left-Arrow	Translate (scroll, pan) to the left, +10 pixels (also Shift + Numpad-4)
Shift + Up-Arrow	Translate (scroll, pan) upwards, -10 pixels (also Shift + Numpad-8)
Shift + Down-Arrow	Translate (scroll, pan) down, +10 pixels (also Shift + Numpad-2)

Arrow	
Shift + Home	Translate (scroll, pan) to the left by 2/3 of the full window width
Shift + End	Translate (scroll, pan) to the right by 2/3 of the full window width
Shift + Page-Up	Translate (scroll, pan) upwards by 2/3 of the full window height
Shift + Page-Down	Translate (scroll, pan) down by 2/3 of the full window height
S	Provides switching between monocular view (standard) and several stereo viewing (binocular) modes . The setting change will be confirmed with a dialog message displayed. <u>Remark:</u> Eventually additional hardware and license extensions might be required. <u>Remark:</u> This setting requires a valid license feature STEREO3DVIEW <u>Remark:</u> This function will have no effect if there are no stereo modes available.

Immersive stereo display rendering techniques

By default 3D renderings are provided for monocular view onto 3D objects projected onto the display (monitor) plane either with or without perspective transformations.

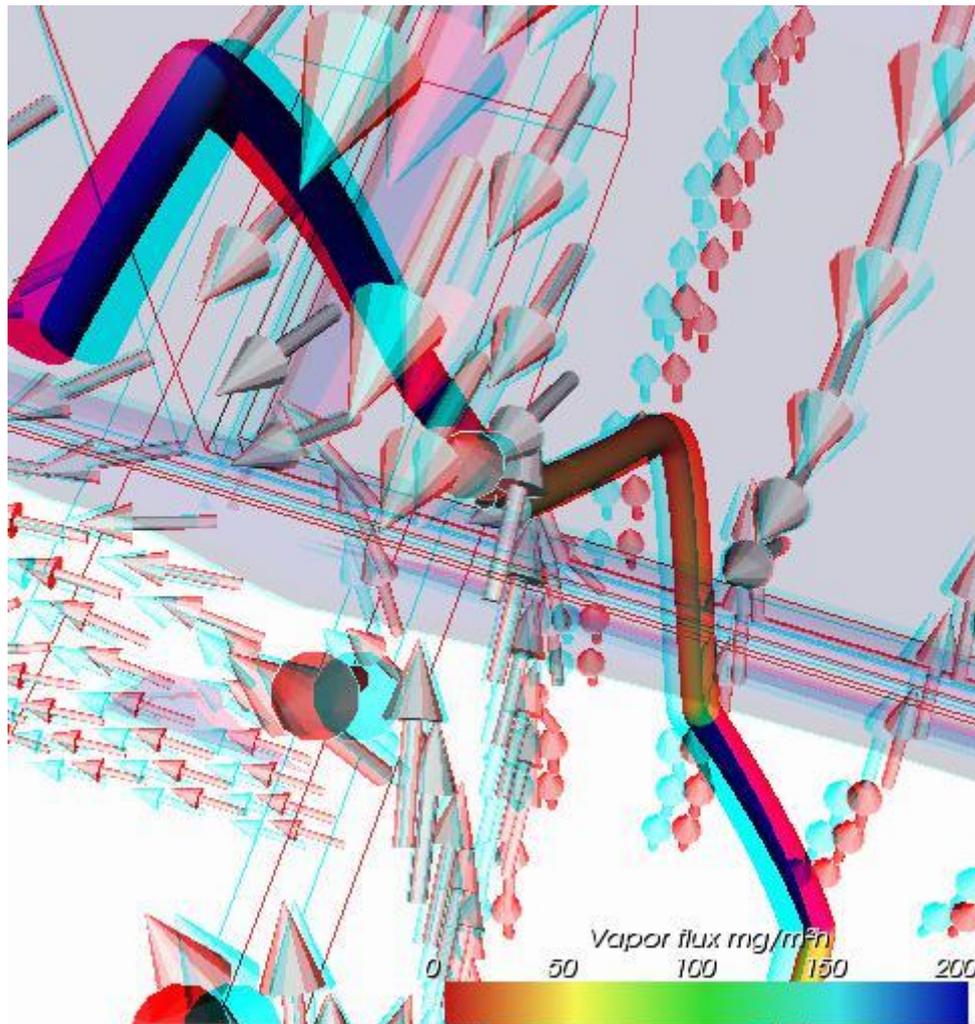
Binocular viewing is accomplished by providing different images to the left and right eye thus allowing for immersive depth perception. Some of the techniques require only very simple additional equipment. Others have more sophisticated technical prerequisites which must be fulfilled to use them.

Implementation provided monocular and binocular rendering modes			
mono (no stereo)	Monocular view (standard).	True Color	No additional equipment required
Anaglyph (Red/Cyan)	By using simple Red/Cyan Glasses the effect of true depth can be accounted. <u>Remark:</u> By adjusting the anaglyph color mask different color filter glasses can be used. The anaglyph color saturation can be adjusted too.	Some Color	Red/Cyan glasses required
Red/Blue	By using simple Red/Blue Glasses the effect of true depth can be accounted at the cost of colour quality (the image will be seen in levels of gray).	Monochrome	Red/Blue glasses required
Dresden	For output to a Dresden-style display.	True Color	Dresden type interlaced display required
VRex (Interlaced)	For output to a VRex stereo projector. All of the odd horizontal lines are from the left eye, and the even lines are from the right eye. The user has to make the render window aligned with the VRex projector, or the eye will be swapped.	True Color	Interlaced VRex type projector required
Checkerboard	For output to a checkerboard style display.	True Color	Checkerboard type interlaced display required
Crystal Eyes	Active stereo-mode display.	True Color	Crystal Eyes type display and active 3D shutter glasses required
Left Eye (mono) Right Eye (mono)	Images shown in mono which shall be directed to the left and right eye respectively can be used to create binocular views based on those two images but in some other stereo viewing application.	True Color	No additional equipment required

STEREO3D-option: Stereoscopic, binocular display of three dimensional evaluations is only possible with an active STERO3DVIEW-Option of the program.



Remark: Activating any binocular stereo modes will have no effect if there are no stereo modes available.



Remark: Only passive stereo view types can be [copied into the clipboard or image file](#). Disabling of stereo mode is only required for hardware dependant stereo types.

Remark (technical): The [anaglyph color saturation factor](#) ranges from 0.0 to 1.0: 0.0 means that no color from the original object is maintained, 1.0 means all of the color is maintained. The default value is 0.65 and can be changed within [application settings](#). Too much saturation can produce uncomfortable 3D viewing because anaglyphs also use color to encode 3D.

Remark (technical): The [anaglyph color mask](#) values are two numbers which are bits mask that control which color channels of the original stereo images are used to produce the final anaglyph image. The first value is the color mask for the left view, the second the mask for the right view. If a bit in the mask is on for a particular color for a view, that color is passed on to the final view; if it is not set, that channel for that view is ignored. The bits are arranged as r, g, and b, so r = 4, g = 2, and b = 1. By default, the first value (the left view) is set to 4, and the second value is set to 3. That means that the red output channel comes from the left view, and the green and blue values come from the right view. This can be changed within [application settings](#).

See also: [Elements 3D](#), [Results 3D](#), [Evaluations and Results](#), [3-D Navigation \(control panel\)](#), [TrackBall Rotation](#)

Trackball and Joystick Rotation

You can adjust the position of the 3D view according to your needs.

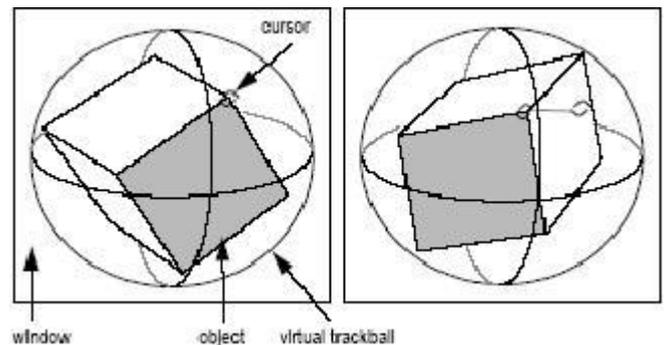
A dragging action (holding down the left mouse button, then moving the mouse) **rotates the view** in 3D space.

The view behaves as if it were attached to a "virtual trackball" whose centre is at the centre of the view window. The mouse cursor is attached to the part of the trackball that protrudes above the surface of the window.

While the [trackball interaction mode](#) is active, the view movements follow movements of the mouse.

While the [joystick interaction mode](#) is active, the distance of the mouse pointer from the centre of the view controls the directions and the speed of continuous movements of the view (occurring while the mouse button is held down, without the need of mouse movement itself).

Remark: You can switch between the two interaction modes ([trackball](#) or [joystick](#)) with [keyboard controls of 3D view](#).



Level Of Detail (LOD) and rendering performance (fps)

While the 3D view is interactively moved with the mouse the application will reduce the level of detail of the displayed entities if required. The desired view update rate (number of frames per second) will be managed as far as possible. As soon as the view stops moving (not being driven with the mouse anymore) the highest rendering quality will be applied again.

The desired view update rate can be set within [application settings](#).

Remark: While considering very complex visualization switching to so called "[Immediate-Mode](#)" application setting will favour memory efficiency instead of speed.

Following objects will be rendered in various levels of detail:

- [Isosurface](#),
- [Model](#),
- [Model slices](#) (all three X,Y,Z),
- [Slices](#) (all three X,Y,Z),
- [Streamline\(s\)](#),
- [Vectors \(HedgeHog, Arrows\)](#)
- [Surface](#)

There are three levels of detail:

- The top level is just the normal data at highest quality
- The middle level of detail is a point cloud of a fixed number of points that have been randomly sampled from objects
- The lowest level of detail is a simple bounding box outline of the object

Remark: Currently a very simple method of TotalTime/NumberOfObjects is used.

Remark: On some graphics systems the point cloud rendering can result in points so small that they can hardly be seen. These hardware dependant behaviour cannot be controlled currently.

See also: [Elements 3D window](#), [Results 3D window](#), [3D Navigation within 3D window](#)

Solid or Wireframe model (setting)

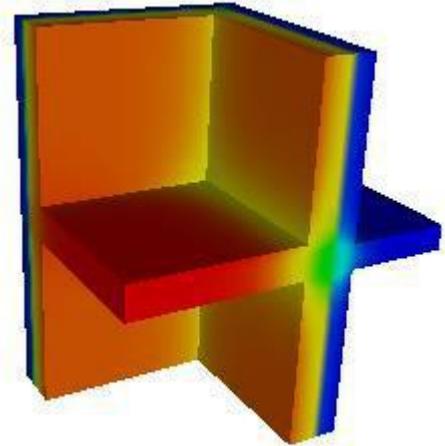
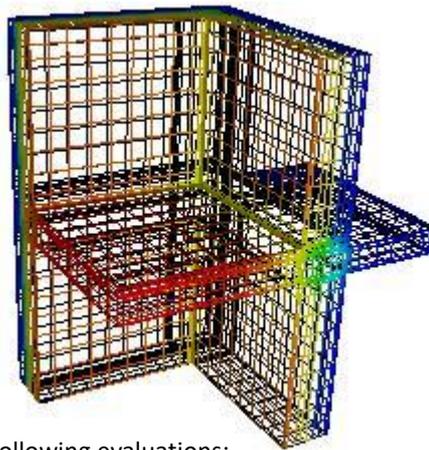
 Solid

By changing the setting "Solid" you can decide if graphical representation is displayed as solid model or as a wireframe (lines) model.

While this setting is turned on, all volumes and surfaces are shown as such.

If this setting is turned off, all volumes will be represented only by their edge lines and surfaces will be shown as a network of lines on top of which surfaces are spanned.

This setting has no impact on how linear or point objects (like component edges, bounding box, axes) and other special object (e.g. colour bar, probe text) are displayed (and is meaningless for such kinds of objects).



This setting can be applied to the following evaluations:

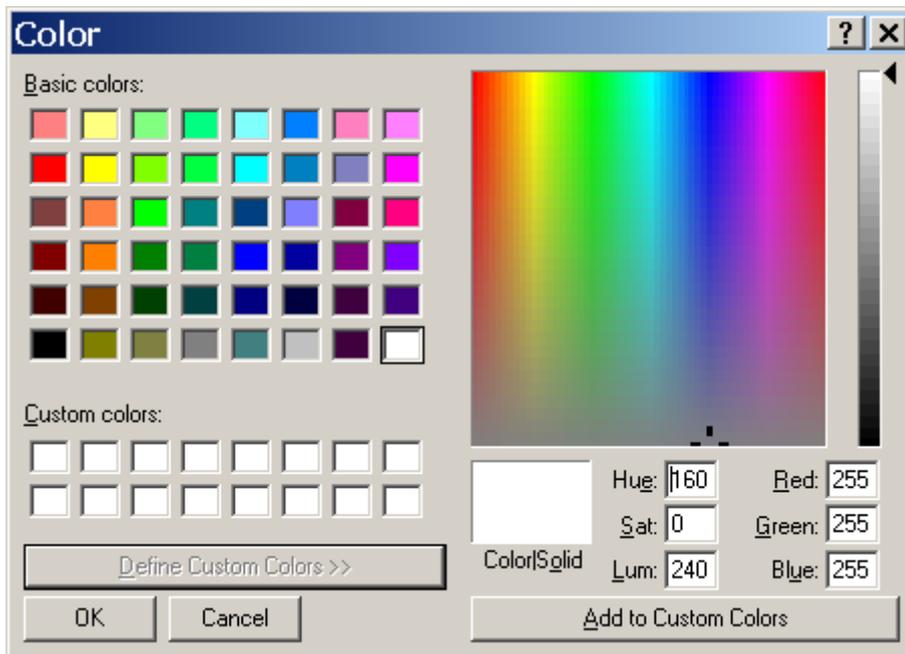
- [Surface](#)
- [Slice X](#)
- [Slice Y](#)
- [Slice Z](#)
- [Streamline](#)
- [Vectors \(HedgeHog, Arrows\)](#)
- [Isosurface](#)

See also: [Elements 3D](#), [Results 3D](#), [Active](#), [Opaque](#), [Colorize](#), [More Light](#)

Background colour (setting)

Back Color

By this setting you control the background colour used in graphical output.



To change the colour of the background click onto the coloured button. Then select the colour from the colour dialog window - a standard operating system specific component.

Select the colour of your choice and confirm your choice with the Ok button.

The background colour can be preset within [application settings](#) also.

Remark: This setting has no impact on the background colour of images saved via the function "Copy Image...". Images placed onto the clipboard with the function "Copy Image..." are always recreated with *white* background colour.

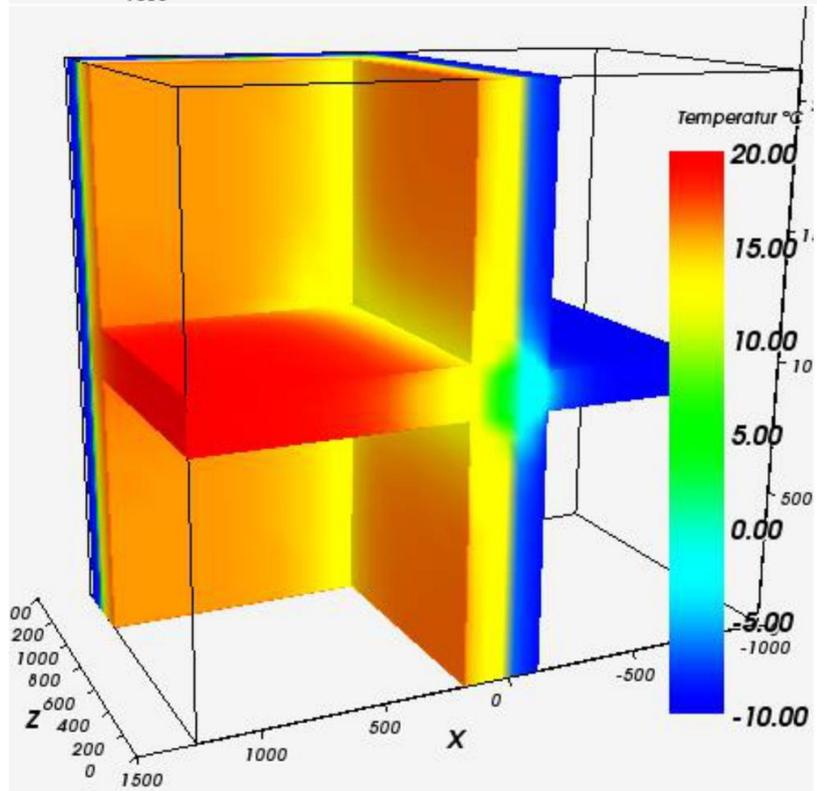
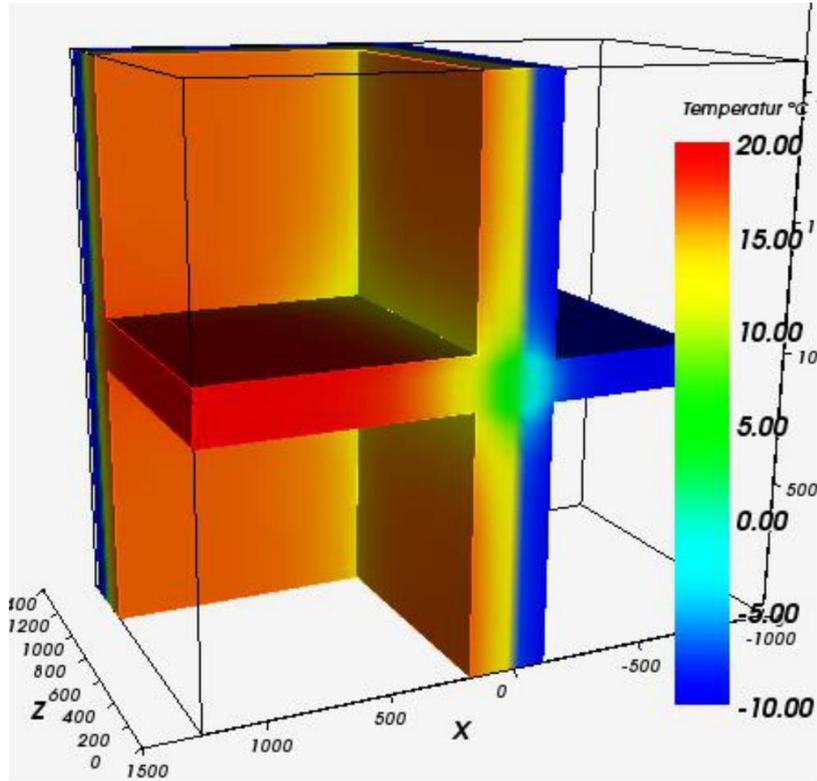
See also: [Elements 3D](#), [Active \(setting\)](#), [Solid or Wireframe \(setting\)](#), [More Light \(setting\)](#), [Application settings](#)

More Light (setting)

More Light

The setting "More Light" is used to turn additional "light sources" on, making the resulting graphical representation look brighter or providing more contrast. Please observe, that by adding more light that way, some areas of the picture might be coloured in much brighter colours. This will be perceived as some kind of colour shift, which, under certain circumstances, leads to an overexposed image.

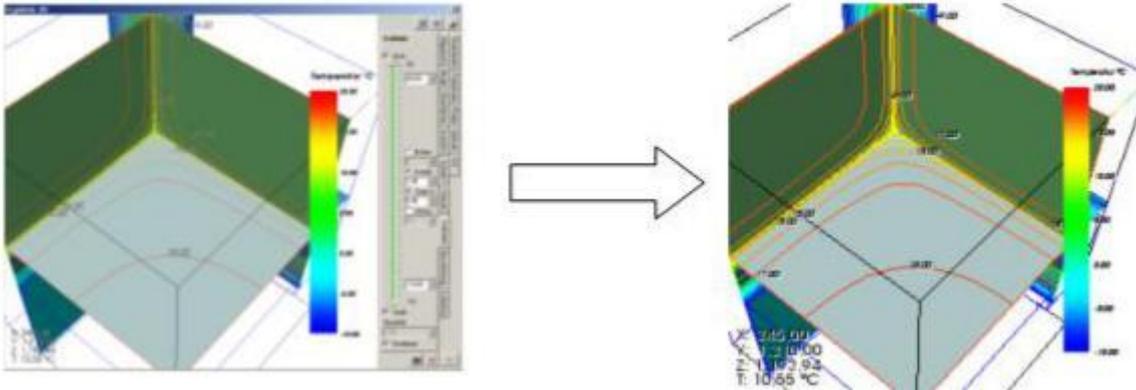
Below you can compare two images created with the setting turned off and on:



Copy image to clipboard (command)

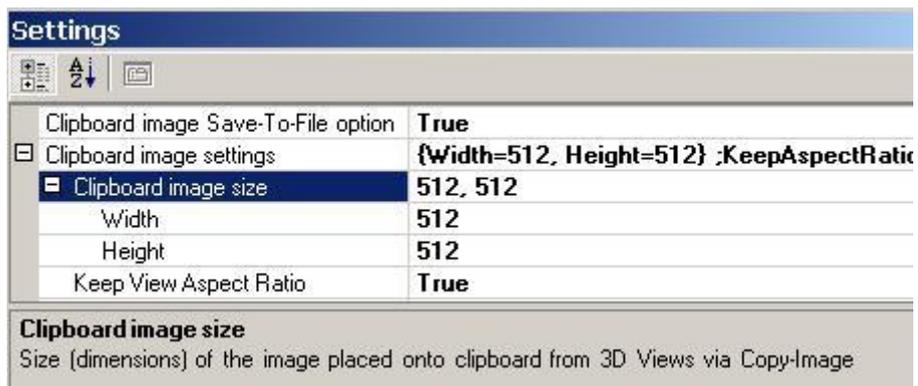


Graphical views can be easily *copied to the clipboard* and transferred for further processing (e.g. for integrating images in reports) to any other application (*pasted from the clipboard*).



Pictures copied to the clipboard are always shown on top of white [background](#) ensuring correct printing quality.

The [size](#) and proportions ([aspect ratio of width to height](#)) of the [specially generated image](#) can be defined within [application settings](#).



During the process of image creation the mouse cursor changes to a sand clock.

Detailed description of clipboard usage is described within operating system help. See [Exporting an image](#) for further description of exporting images from this application.

If the [application setting "clipboard image Save-To-File option"](#) is set to active (true) then you will be offered an option to save (export) the image into an [image file \(PNG, TIF, BMP, GIF, JPG\)](#) - a *file-save-as-dialog* will be presented.

Remark: The option to save images placed onto the clipboard to a file can be set within [application settings](#) (Clipboard image Save-To-File option, [AlwaysAskToSaveGrabbedImage](#)).

Remark: The meaning of the setting "Clipboard image Save-To-File option" can be reversed while the SHIFT-key is held depressed during a click onto the "Photo button". This means, that if the Save-To-File option is active saving the image to the file will not be offered if the Photo-button is depressed during the SHIFT-Key is held depressed. Opposite is also valid: During the Save-To-File option is not active (set to False) and the Photo-button is clicked while holding the SHIFT-key, the dialog offering saving of the image to a file will be presented in contrast.

Remark: The [3D cursor widget](#) and the [orientation marker](#) will be hidden temporarily and not rendered to the resulting export view.

Remark: Saving images while [in stereo mode](#) might be eventually restricted (see "[stereo display rendering](#)").

See also: [Elements 3D](#), [Results 3D](#), [Application settings](#), [Exporting an image](#), [Keep view aspect ratio](#)

Working with files

The software saves data entered by the user as well as calculated intermediary results into different, project specific, files and folders.

Project files

Data of each project is stored in so called "*project files*". For all *project files* the software expects and uses the file name extension ".*antherm*". It is under users' control choosing some different extension for certain purposes instead of the extension (".*antherm*") suggested by the program.

The name of the project file must be unique within the folder it is saved into. Of course it is allowed to have project files with equal names saved to different folders.

For the purpose of **Data backup** it is absolutely sufficient to make secure copies of project file. All data and settings required to reproduce any calculation results (even if requiring complete recalculation) and all project specific parameters are kept in the project file.

Project file stores following data - provided this data has been entered:

- description of the project,
- materials,
- surfaces,
- elements or layers and its elements,
- registration parameter of an image underlay (optional)
- fine grid parameters,
- solver parameters,
- boundary conditions (including periodic data for each main period of transient calculation)
- probe points,
- isometries,
- results3D parameters.

The file name extension ".*antherm*" is signalling to the operating system, that it is the input file for AnTherm. A double click onto an AnTherm project file [starts](#) the application and loads the chosen file.

Remark: Also [dropping \(drag & drop\) a project file over the main application window](#) will open that file.

Opening or saving of project files is executed from the [menu "File"](#). All project data is also saved automatically at the beginning of each new [calculation run](#) or new evaluation. At any time project data can be saved by choosing the menu [File→Save...](#). This is especially useful during evaluation to save current boundary condition for example too..

The software will offer the option to save data prior to any potentially destructive operation. This "safety" behaviour can be turned off in [application setting](#) by changing the value of the option "[always ask to save project](#)".

It is recommended to save project data before executing following actions:

- [New project \(2D, 3D, Schichten 3D\)](#),
- [Load, Load resent](#),
- [Import](#) (waebru 2/3bt, acad/DXF, Heat2, Heat3, Kobru86)
- [Convert](#) (to 2D, 3D, layered 3D)
- [Exit](#) / closing the main window

Remark: During the initial save operation the application will create the [standard project folder](#) and offer this in respective dialog as central storage point for project files. The name of this folder is defined by the application setting "[Default project directory](#)".

Important: The overall path length of a project file (i.e. including drive and folder names $L:\backslash\text{folderpath}\backslash\text{projectfile.antherm}$) is constrained by operating system limits. When the current limitation of Windows is taken into account together with path lengths required for the [folder of simulation results](#) and its contents the maximum path length of a project file must not exceed 220 characters. Simulation results will not be created if the path name is longer then 220 characters - the project has to be saved under different name or into different folder (with overall path length not exceeding 220 characters) to be able to obtain simulation results also.

Folder of simulation results

For each project AnTherm keeps *tentative calculation results (simulation results)* of calculations stored in a subfolder "*<projectfile>.dir*".

This means, that for a project file named *<name>.antherm* there will be a subfolder *<name>.antherm.dir* storing simulation results. The project file and the related subfolder of simulation results accompany each other and therefore they share the name.

The absence of a simulation folder signals, that there has been no simulation run yet. If the folder *<name>.antherm.dir* has been deleted by the user then, on the next request to show results of the project *<name>.antherm*, complete calculation run will be executed. Deleting the simulation folder can therefore be used to force full recalculation. Since all data required for reproduction of simulation results along with all project specific settings are kept in the project file, the loss of simulation folder does only mean, that additional time is required to repeat all the required computations.

Contents of the simulation folder "*<projektdatei>.dir*" are fully managed by AnTherm - without any intervention required from user. The simulation results folder has special, application specific, structure and thus it is not designed to be accessed by the user directly. AnTherm creates this folder and, under certain circumstances, will purge its contents also without warning. For that reason no user specific files or data shall be placed into the folder "*<projektdatei>.dir*" ! Manipulating contents of the simulation folder might also break its data integrity leading to serious software malfunction - therefore it is forbidden!

To avoid accidental manipulation AnTherm applies hidden attribute to each simulation results folder. This makes this folders invisible and unavailable for browsing within windows file explorer by default. Only if the user changes the setting "show hidden files and folders" within windows file explorer, simulation results folder will be shown therein. This specific setting of windows file explorer does also apply to results shown in the dialogs "Open project..." and "Save project".

Note: Simulation results of a vapour diffusion computation also stored as a subfolder *DAMPF* within the simulations results folder "*<projektdatei>.dir*".



For the purpose of **Data backup** it is under users control to save simulation results folder also - in addition to project file. If the project file *<name>.antherm* and its accompanying simulation results folder *<name>.antherm.dir* are together restored, the application has no further need for full computation of simulation results.

The [Simulation folder cleaner](#) tool supports you in removing (physically deleting) unneeded simulation results folders to save disc space.

Important: Calculation of a project with its name given by a UNC name (e.g. /server/path...\project.antherm) is not allowed. If the given UNC path name cannot be coupled with a temporary network drive the application will show explaining message accordingly. In general the calculation of a project on a network drive (connected by a drive letter) is possible but (due to significant performance penalties) not advisable.

Variant calculation, renaming and copying of project file

Under certain usage scenarios there might be the need to execute calculation of variants which differ only slightly by few parameters. In such a case it is the easiest to simply save the project with new name. However this results in the need of complete simulation calculation because the simulation folder does not exist yet for the new project name.

Remark: The need for performing complete new simulation calculation results when significant input data or parameters of a project have changed: e.g. geometry of the construction, material and/or surface properties, names (tags) of spaces, power sources, materials or surfaces, fine grid parameters etc. This concludes that only under very rare occasions there will be no need for complete new simulation calculation to be executed.

Still to avoid repeating simulation calculation when performing variant modifications one can copy the project file and its accompanying simulation results folder with simple tools of the operating system (e.g. windows file explorer, command line etc.):

Variant scenario A:

Use subfolders to store variants: copy the project file *<name>.antherm* and its accompanying simulation folder *<name>.antherm.dir* into a new folder. That way you receive a subfolder "Variant B" containing equally named copy of already calculated project and its simulation results from folder "Variant A".

Variant scenario B:

Make a copy of the project file `<nameA>.antherm` and name it `<nameB>.antherm`. Make a copy of the simulation folder `<nameA>.antherm.dir` also and name it `<nameB>.antherm.dir`. That way you have an exact copy of project "nameA" and its simulation results folder, now both named "nameB".

Miscellaneous files

Beyond project files the application stores further information in additional, application specific, files:

- [Standard project template file projekt.xml](#)
- [Initial template of window positions AnTherm.exe.LayoutPersistence.xml](#)
- [Settings file AnTherm.exe.Settings.xml](#)
- [Colour setting file LambdaToColor.ColorList](#)
- [Material data base file Baustoffe.xml](#)
- [License file AnTherm.HID](#)

This application is aware of importing files created with other applications:

- [Import of Image file \(Image Underlay\)](#)
- [Import of DXF file](#)
- [Import of WAEBRU component file](#)
- [Import of Heat2 input file](#)
- [Import of Heat3 text input file](#)
- [Import of Kobru86 file](#)

There are some [example project files included with the application's installation](#) which can be used to execute tutorials and learn the program's use. These example project files can be found in the subfolder "Beispiele" (means "*examples*" in german) of the installation folder (typically `C:\Programm\tkornicki\AnTherm\Beispiele`) or can be directly loaded into AnTherm via the menu [Help->Example Load...](#) .

Results created with the application can be exported for further processing:

- The matrix of thermal coupling coefficients (conductance matrix, Leitwertmatrix) can be exported to a [CSV file](#),
- All reports can be saved as [PDF, XLS, DOC or RTF file](#),
- Graphical representations can be [saved as picture file \(PNG, TIF, JPG, BMP, GIF\)](#),
- Graphical representations can also be [transferred as pictures via clipboard to other applications](#),
- Graphical representations can be exported [3D scene file \(VRML, IV, OOGL, ...\)](#).
- Animations can be [recorded as video files \(AVI\)](#).

See also: [The main menu](#), [Tutorials and cookbooks](#), [Simulation folder cleaner](#)

User specific Logo CustomColorBitmap.bmp

A user specific logo picture can be set in the [application settings](#). The preset value uses an example file *CustomLogoBitmap.bmp* from the installation folder by default.

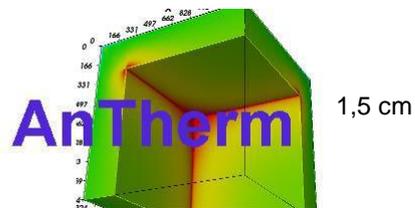
The logo image is displayed in the heading of reports, as middle item of the heading, placed left to the Copyright-Info, 6cm wide and 1.5cm high.



Any picture file can be used (format BMP, GIF, JPG,...).

To avoid distortion of the picture keep the width to height ration of 4:1 (for example 800x200 pixels).

6,0 cm



If one wishes that no logo picture is displayed on reports, simply leave the setting empty.

Important: Under certain circumstances the logo picture will be overlapped by the licensing information placed left to the logo. The example file "CustomLogoBitmap.bmp" shows the solution containing the logo itself placed on the right edge of the bitmap and allowing the left side of the image, which has been left in the background colour of white, to be overlapped by the license information by intent.

See also: [Application settings](#), [Evaluation reports](#)

Standard project template file projekt.xml

If the installation folder contains the file named *projekt.xml* and [application settings](#) chosen are not set to automatically reopen the latest project, this template project file will be automatically loaded on applications start.

One can use this file to create your own "template project" containing some preset materials or text input to have them at your hand for new projects.

This behaviour can be changed via menu function [application settings](#) directing AnTherm to automatically reopen the latest project instead of this template file.

Remark: The preinstalled template file *projekt.xml* contains some example project and shall be modified by the user.

See also: [Application settings](#), [Working with files](#)

Initial template of window positions AnTherm.exe.LayoutPersistence.xml

On the absolutely first start of the application its windows will be shown at some tentative positions on the screen. These initial positions are defined by the pre-installed template file "AnTherm.exe.LayoutPersistence.xml" which can be found in the installation folder. The tentative layout provided assumes screen resolution of 1024x786 pixels.

AnTherm will only read the file „AnTherm.exe.LayoutPersistence.xml“ is the application settings file [AnTherm.exe.Settings.xml](#) does not exist yet. During the application shutdown AnTherm saves window positions to the user specific settings „AnTherm.exe.Settings.xml“ and on subsequent executions only this file is being used!.

Resetting the window positions to its initial, tentative, setting can be accomplished with the menu function [Windows→Reset window positons](#).

The format of this template file conforms to the section "LayoutPersistence" of the application settings file "[Antherm.exe.settings.xml](#)".

See also: [Application settings](#), File [Antherm.exe.settings.xml](#)

Colour setting file LambdaToColor.ColorList

The setting used for automatic colouring of material cells based on its thermal conductivity is kept in the file *LambdaToColor.ColorList*.

The file can be found in the installation folder and is being automatically loaded at application start up.

This text file starts with a header line (predefined content) followed by list of colours (ARGB Bytes) in sorted intervals of conductivity values (Double).

The colouring according to the defined intervals of thermal conductivity (lambda) can be redefined by editing this file..

See also: [Application settings](#), [Working with files](#), [Element Editor](#)

Import of WAEBRU component file

A component file created with the application package WAEBRU V6.0 can be imported into AnTherm and automatically converted into AnTherm project file nearly without any changes.

Please observe following remarks:

- A .2BT file (a two dimensional WAEBRU model) will be converted to AnTherm's [2D project](#).
- A .3BT file (a three dimensional WAEBRU model) will be converted to AnTherm's [layered 3D project](#).
- If there is a raster-file corresponding to the imported WAEBRU component available within the same folder (name.2RA or name.3RA), rasterising parameters used to create this file (and stored within it) will be imported also and converted to [fine grid parameters](#) of the AnTherm's project.
- Each "power source" element of a WAEBRU model is converted into two respective elements of AnTherm. The first one as a material element and the second element as power source extent of same dimensions - overlapping the corresponding material element.
- Names of component groups and components assigned to elements within the WAEBRU file are converted to group assignments of each element.
- Element coordinates stored within WAEBRU file are always read rounded at **third decimal position**.
- Project description is transferred unchanged from WAEBRU file.
- Reinforcing bars (neither diagonal nor parallel to axes) will not be imported in current version!
- During the import of a WAEBRU file <name>.BT AnTherm will also attempt importing a corresponding vapour model from a subfolder DAMPF (created with MueInp - an internal DAMPFBRU tool) thus loading the vapour transfer coefficients (μ) also. 
- If the imported WAEBRU file originated from an instationary (transient, harmonic) application variant of WAEBRU (iwaeburu), AnTherm will import heat capacity and mass density (ρ and c) properties of materials

Remark: Some imported WAEBRU component files can contain coordinate values saved with rounding errors due to (that time common) number representation. To compensate for such rounding errors AnTherm will always round coordinates read from the WAEBRU file at third decimal position (WAEBRU allows entry of only to decimal places, but it writes four decimal places to the file saved). Following arguments of the WAEBRU file are concerned: Ex1, Ex2, Ey1, Ey2, Edis, BGDicke.

Remark: If the import from a WAEBRU file is not successful an error message will be shown. The resulting AnTherm project will be empty.

Remark: A WAEBRU file can be dragged from windows file explorer and [dropped over the main application window](#) as a shortcut for quick import.

See also: [Working with files](#)

Import of Heat2 input file

Components available as stored within HEAT2 input file can be easily imported as used as starting point for further modelling and simulation.

Only material elements and empty boxes are imported from a Heat2 input file.

Remark: An import of a Heat2 input file always creates a [2D project](#).

Remark: The volumetric heat capacity will be imported into the Rho value. Ce will be set to 1.0.

Post processing of imported data

Because only geometry and conductivities of material elements are read from a Heat2 input file, but no boundary conditions (spaces, power sources), the project created by importing a Heat2 input file must be edited further prior to simulation.

Post processing steps:

- Enter project description.
- Creation of spaces. Location of spaces connected to the model must be defined manually.

Hint: The quickest possibility to add spaces is by inserting space elements at the top of element list, thus letting all other (imported) elements to overlap them accordingly.

See also: [Working with files](#), [Editing windows \(input windows\)](#), [Element selection window](#)

Import of Heat3 text input file

Components available as stored within HEAT3 text input file can be easily imported as used as starting point for further modelling and simulation.

Only material elements and empty boxes are imported from a Heat3 text input file.

Remark: An import of a Heat3 text input file always creates a [3D project](#).

Remark: The volumetric heat capacity will be imported into the Rho value. Ce will be set to 1.0.

Post processing of imported data

Because only geometry and conductivities of material elements are read from a Heat3 text input file, but no boundary conditions (spaces, power sources), the project created by importing a Heat3 text input file must be edited further prior to simulation.

Post processing steps:

- Enter project description.
- Creation of spaces. Location of spaces connected to the model must be defined manually.

Hint: The quickest possibility to add spaces is by inserting space elements at the top of element list, thus letting all other (imported) elements to overlap them accordingly.

See also: [Working with files](#), [Editing windows \(input windows\)](#), [Element selection window](#)

Import of Kobru86 file

Components available within Kobru86 file (e.g. from the KOPRACTICE data set) can be easily imported as used as starting point for further modelling and simulation.

Only material cells will be imported from a Kobru86 file. Position of imported space boundaries must be verified manually after import.

Remark: The importing procedure uses text and material files (if contained within the directory structure) also. Two files, KOBRSTRI.TXT and MADB.TXT are looked for at each import within the directory at which the Kobru86 file is located. If either of this files cannot be found within this directory the search for these file continues in the parent directory.

Remark: An import of a Kobru86 file always creates a [2D project](#).

Remark: If the text file KOBRSTRI.TXT (space names), corresponding to the thermal heat bridge imported, cannot be found, AnTherm will set space names to "KOBRSTRITXT_LN=lineNo".

Remark: If the text file MADB.TXT (properties of materials), corresponding to the thermal heat bridge imported, cannot be found, AnTherm will set material names to "MADBTXT_RN=refNo".

Post processing of imported data

Because importing of boundary conditions (spaces) from Kobru86 file is not fully automated, the project created by importing a Kobru86 file must be edited further prior to simulation.

Post processing steps:

- Enter project description.
- Creation of spaces. Location of spaces connected to the model must be verified and eventually adjusted manually.
Hint: The quickest possibility to add spaces is by inserting space elements at the top of element list, thus letting all other (imported) elements to overlap them accordingly.

See also: [Working with files](#), [Editing windows \(input windows\)](#), [Element selection window](#)

Import of DXF file

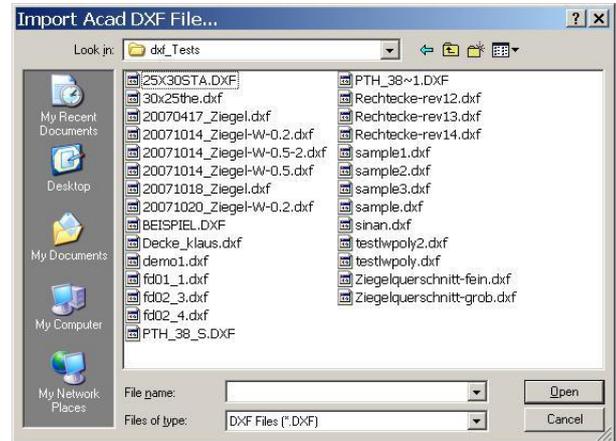
If there is a construction available as a DXF file (e.g. created with AutoCAD® or other CAD application), it can be easily imported and used as a template for creation of a [two dimensional component model](#).

Only [specifically prepared DXF files](#) can be imported. Only if a DXF file conforms to [specific criteria](#) it can be imported into AnTherm.

To start the DXF file import, select the menu [File→Import→Acad-DXF](#).

The file browser shown provides the ability to search for DXF files. Select the DXF file to import and press "Open" or double click onto the file requested.

The DXF file is read in and all polylines are processed. Prior to actual conversion you will be shown an import settings dialog. The import settings dialog is used to control how the contents of the DXF file imported are interpreted by AnTherm. You shall define by which attribute materials or conductives shall be identified: either by layer name, the colour or the line respective polyline. If no specific material grouping defined during creation of the DXF file, select type of each has been "ignore".



The dialog displays the count of polylines adequate for import (after the refine-step) and corresponding layer-, line- and colour names which shall be used for either material name assignment or grouping (if there are no line types or layer names defined within the DXF file "NoLineType" or "NoLayerName" is used).

If the DXF file has been created with component grouping in mind also, then you can select this criteria within the block "group by".

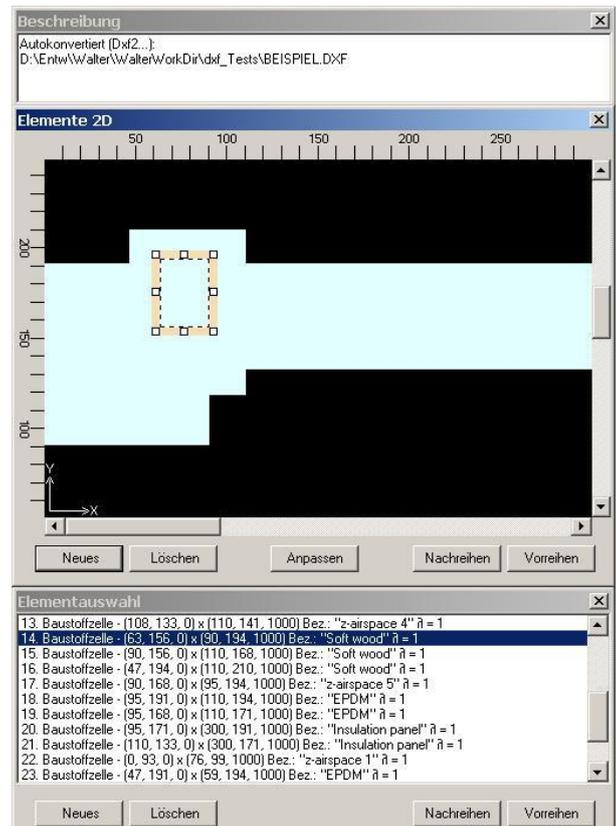
The setting "optimize number of elements" shall be left checked in most cases. The number of elements resulting from the conversion and transferred to AnTherm will be kept as small as possible by merging resulting elements if applicable.

Check the field „Group polygons too“ only, and only then, if component areas surrounded by polylines within the DXF file shall be interpreted as component groups also.

Select the lowest number of decimal places sufficient to precisely read and interpret the geometry saved within the DXF file.

Press "Ok" to initiate the import of polylines and convert them to elements.

Concluding the import of geometry data you will be asked if material properties of materials currently stored in the material's list shall be [applied \(replaced\) by name on all elements](#) of the imported model.

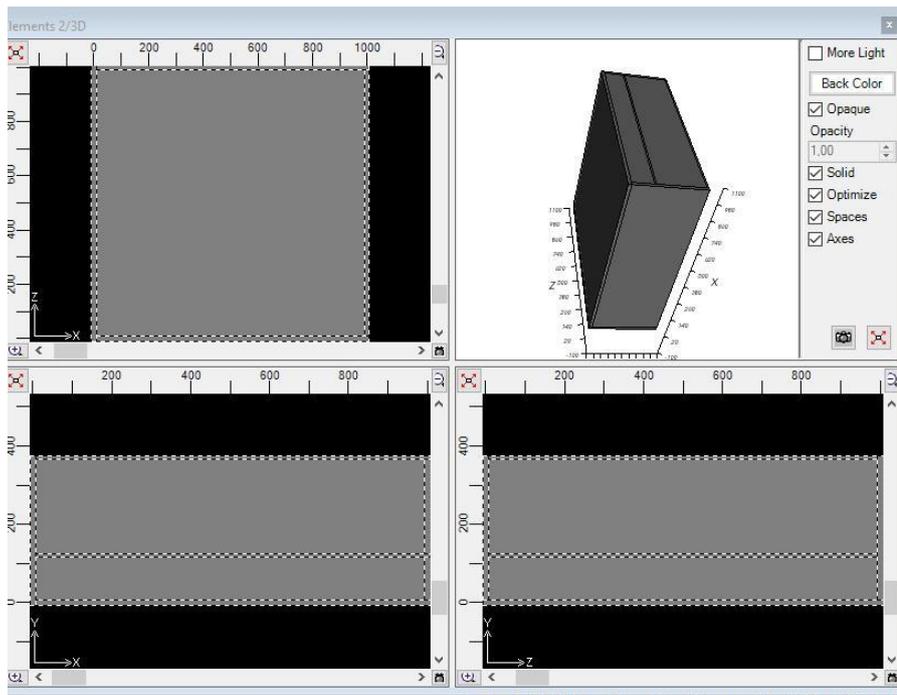
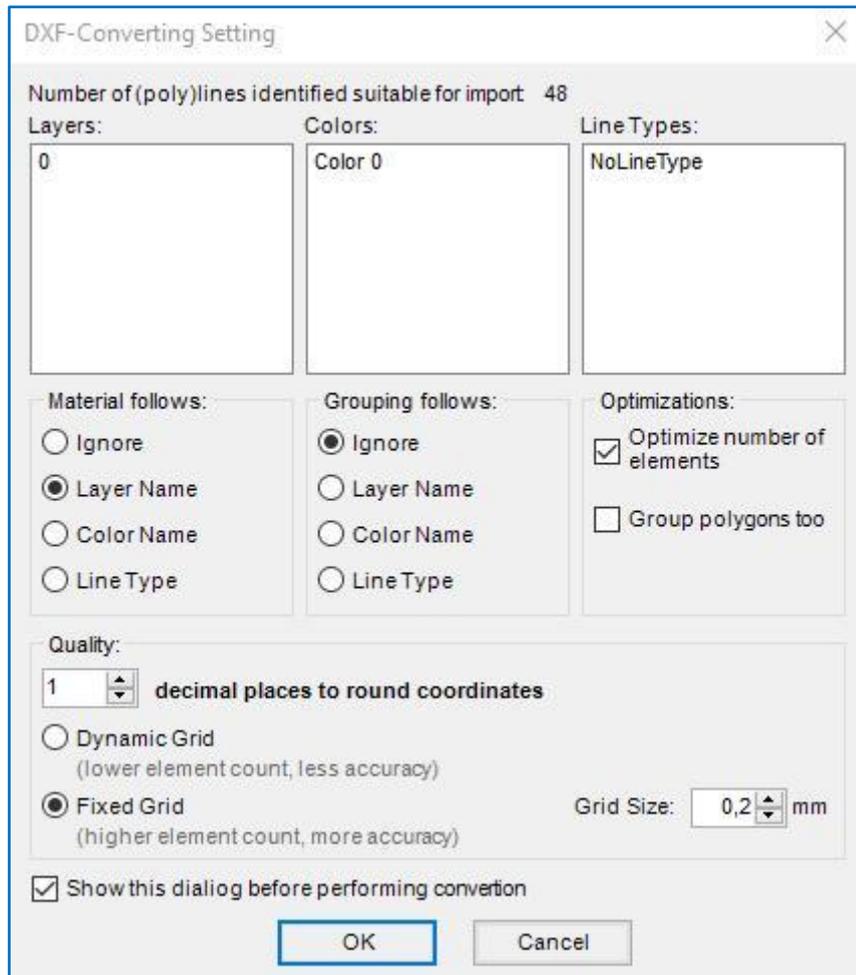


Remark: One shall also include spaces, power sources etc. within the file to be imported and change the type of resulting elements later.

AB Version 10 you have the possibility to import several 2D-DXF files into an existing project. Therefore position the imported elements by using "Transform / Move", „Rotate / Window“.

DXF-Converting Setting

File-Import



Post processing of imported data

Because the imported DXF file provides only geometry data and naming of materials and groups, the data of the imported model must be post processed prior to further computations.

Post processing steps:

- Enter the text of project description.
- Define spaces (boundary conditions).

Typical CAD drawings contain only the component but lack location of attached spaces. Location of spaces connected to the model must be defined manually.

Hint: The quickest possibility to add spaces is by inserting space elements at the top of element list, thus letting all other (imported) elements to overlap them accordingly.

- Entry of heat conductance for materials.

During import only material names are provided by the DXF file.

See also: [Replace Materials on All Elements \(within the Materials window\)](#)

Remark: The order in which polylines are stored in (and thus also imported from) DXF file does not follow the order they have been input within CAD application.

Due to the fact, that elements imported will follow the order of polylines as they are stored within the DXF file, we advise you **not** to use [element \(nor polyline\) overlapping!](#) If necessary you will make the use of [reordering functions \(promote or relegate\) of elements](#) to reach needed overlapping effects.

Remark: If the import from a DXF file is not successful an error message will be shown. The resulting AnTherm project will be empty.

See also: [Requirements related to DXF file, Element selection window, Materials window, Editing windows \(Input windows\)](#)

Import of DXF file - Requirements

If there is a construction available as a DXF file (e.g. created with AutoCAD® or other CAD application), it can be easily [imported](#) and used as a template for creation of a [two dimensional component model](#). Only specifically prepared DXF files can be imported. Only if a DXF file conforms to specific criteria it can be imported into AnTherm.

Below you will find all requirements to follow when creating DXF file which shall be imported to AnTherm.

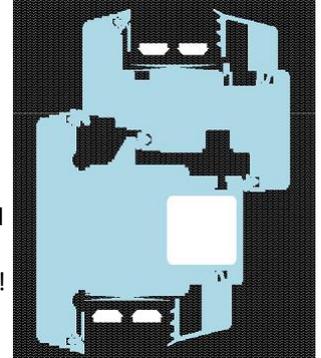
AnTherm 8.132

In the new version the DXF import function was enhanced. Now slopes and roundings are imported too.

Regarding the preparation of 2D projects:

- Polylines no longer have to be axis-parallel.
- Simple lines will also be used if they can be combined to polylines.
- It is also possible to use circles, ellipses and arcs.
- Polylines whose layer name starts with "Space" or "Raum" are automatically interpreted as space elements.

The other requirements are like in previous versions (read on). Note: We do not support hatches!

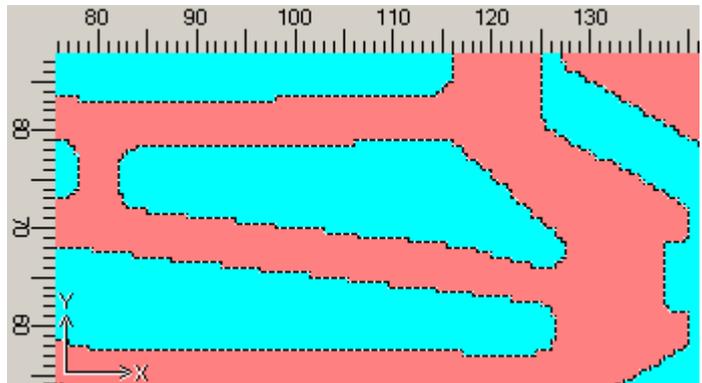


Requirements related to contents of DXF

The application converts contents of a DXF file into a building component. The substantial characteristics of a building model is described in terms of unique assignment of heat conductivity to specific areas of the construction. This information is usually not entered within CAD input. Therefore the DXF file prepared for the import must **rigorously follow some stringent rules**.

With regards to the contents of a DXF file following applies:

- **Only polylines** are read. Any different content of DXF file (lines, points, labels, blocks, hatches etc.) will be skipped during reading.
- **Only closed polylines with segments parallel to axes** are processed. Any polyline containing sloped segments, as well as polylines not declared as closed, will be dropped. (**Exception:** If the first and last point of an open polyline coincide exactly by their coordinates, the application will implicitly close the polyline and consider for further processing.) **Hint:** Polylines within DXF file shall not self intersect! Evaluation of self intersecting polylines will result in unexpected results.
- Assignments to **layers, line types and colours of polylines** will be interpreted specifically by the application.



Polylines assigned same layer number will be interpreted as belonging together in such a way, that areas surrounded by them are filled with same material (hence have same thermal conductivity). The layer name will be used as name of material.

Analogically same applies to polylines of equal line type or equal colour. Such grouping by material can be selected either layer number or line type or colour however. Mixing of the grouping criteria is not allowed!

Hint: Applying some naming scheme which follows a specially prepared [material list](#) consequently, gives you the ability to automatically amend physical properties of materials - see [Replace Materials on All Elements \(within the Materials window\)](#)

Selective use of layers, line types or line colours provides additional ability to generate assignments to component groups. Polylines with equal layer number, equal line type or equal colour can be assigned collectively to one component group. The name of a layer, line type or colour will be used as the name for the group.

Of course it is important to carefully use layer numbers, line types and line colours to create meaningful and helpful groupings. It is advised to use the layer information for the purpose of assignment of polylines to materials and - if required and applicable - use line type as grouping vehicle to generate group assignments.

- While creating the DXF file keep in mind, that CAD drawing units will be **interpreted as millimetres** by the application.
- Coordinates stored within DXF file will be read as **double precision**. This provides the maximum precision of 13 to 15 decimal digits for coordinates read.

Therefore it is advisable to **save coordinates to a DXF file by specifying the smallest number of digits** required to represent precise coordinates (typically 7 to 10).

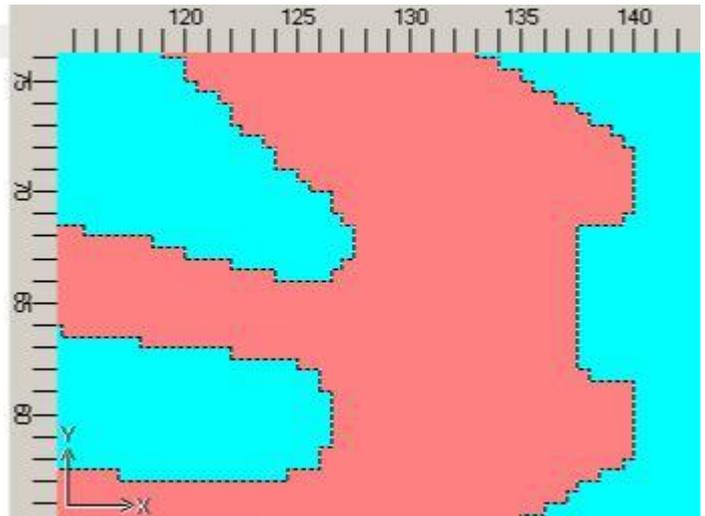
Important: We strongly recommend to avoid writing the maximum of 16 digits as offered by default by some programs!

Creating a DXF file suitable for the import

Resulting from the above considerations following guidelines result for the creation of new or processing of existing DXF files :

Creation of a new DXF file

- Exclusive use of closed polylines with segments parallel to coordinate axes.
- Slopes and curves must be modelled by (replayed with) stairs (with line segments parallel to coordinate axes)
- Grouping of polylines by materials (thermal conductances) either by equal layer number or by equal line type or by equal line colour. (if [some materials list shall be applied](#), stick to the naming convention chosen)
- Grouping of polylines to generate component groups (if required) either by equal layer number or by equal line type or by equal line colour.
- Cleanup the drawing unused elements (e.g. apply "_purge" command in Acad).
- During saving to DXF file request scaling in millimetres.
- During saving to DXF file request the smallest number of decimal digits sufficient to represent coordinates precisely.



Editing of existing DXF file

- Draw over the CAD picture with polylines.
- Slopes and curves must be modelled by (replayed with) stairs (with line segments parallel to coordinate axes)
- Group polylines by materials.
- Group polylines to create component groups also (optional).

Remark: The order in which polylines are stored in (and thus also imported from) DXF file does not follow the order they have been input within CAD application. Due to the fact, that elements imported will follow the order of polylines as they are stored within the DXF file, we advise you **not** to use [element \(nor polyline\) overlapping!](#) If necessary you will make the use of [reordering functions \(promote or relegate\) of elements](#) to reach needed overlapping effects.

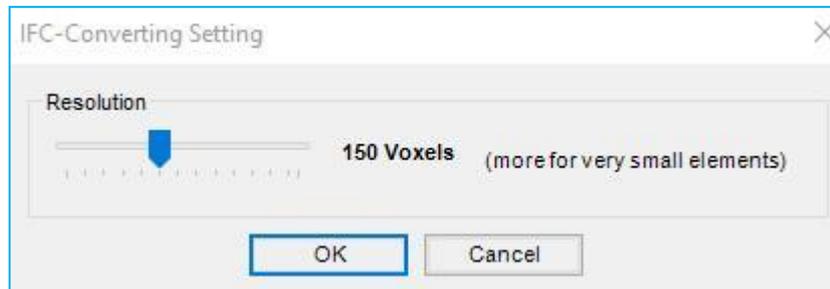
Remark: Different line ending conventions shall be tolerated within import: Windows (CR LF) - Macintosh (CR) - Unix (LF).

See also: [Import of DXF file](#)

Import of 3D DXF / IFC file

AnTherm 8.133 or higher is able to import 3D models in DXF or IFC format that have been created using programs such as AutoCAD, ArchiCAD, Revit,...

Voraussetzung, um diese Funktion benutzen zu können, ist die **Lizenzoption IMPORT3D**.



In order to import a 3D DXF file to AnTherm, the same procedure as for 2D DXF files is to be applied, that is the command File → Import → aCad DXF. For IFC files the command File → Import → Revit IFC is to be used.

The DXF / IFC files need to meet several requirements:

- DXF files must not use the 3DSOLID command. It is possible to force AutoCAD to avoid this command by choosing an old variant of the DXF format (as old as possible) when saving the file.
- 3D objects should be modelled as closed polylines or polyfaces in DXF files so that they can be imported with AnTherm.

In IFC files, AnTherm searches for the command IFCRELCONTAINEDINSPATIALSTRUCTURE. There AnTherm retrieves the list of objects to be displayed. AnTherm is capable of processing several types of objects, for instance

- IFCBUILDINGELEMENTPROXY,
- IFCBUILDINGSTOREY,
- IFCWALL,
- IFCSLAB,
- IFCCOLUMN,
- IFCBEAM

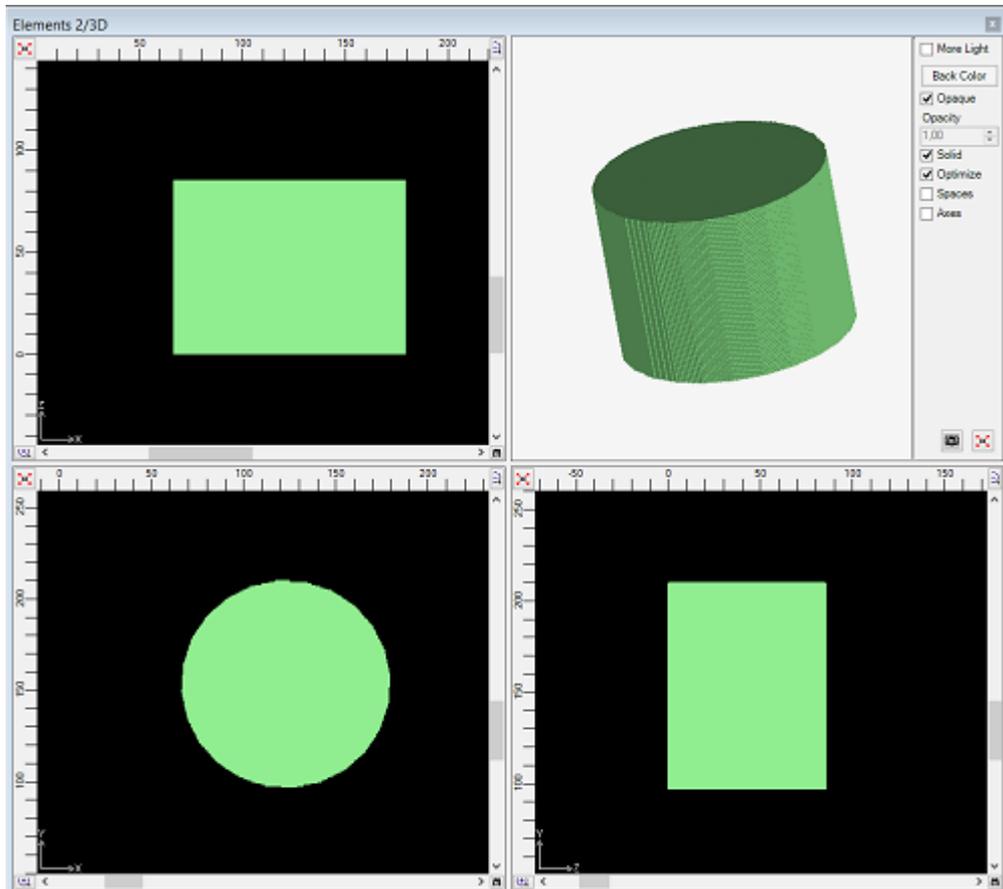
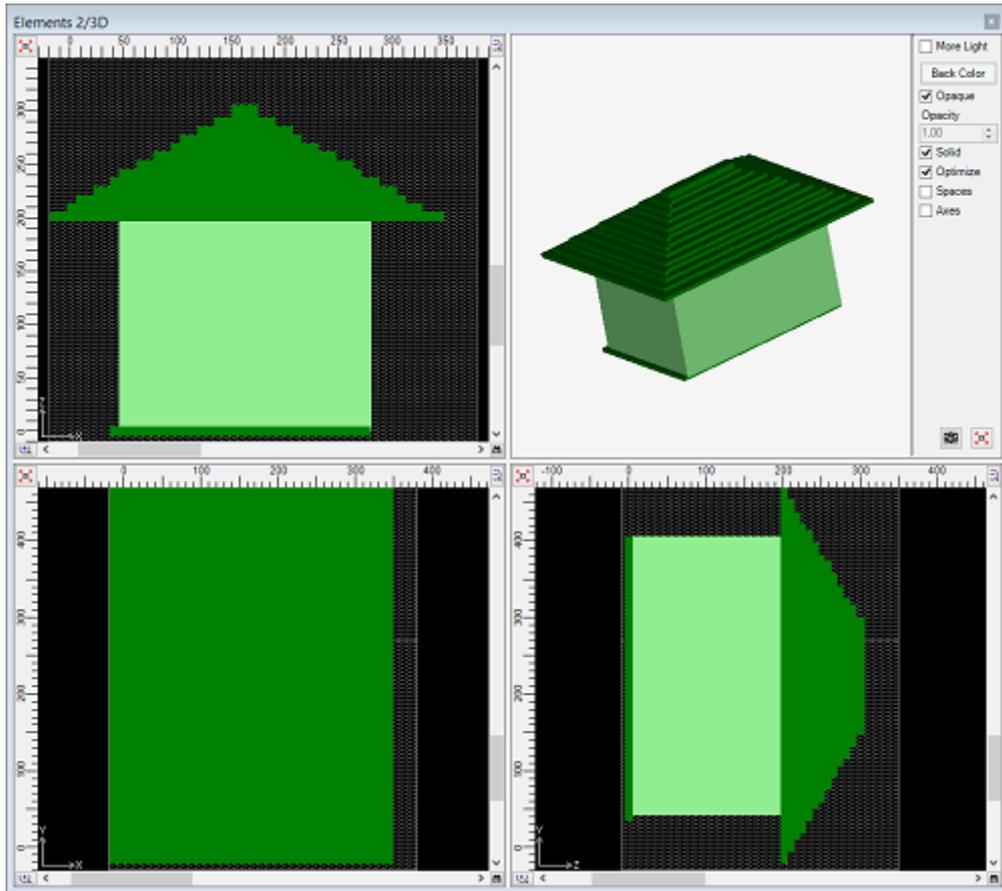
and many more. These objects can be defined using commands such as

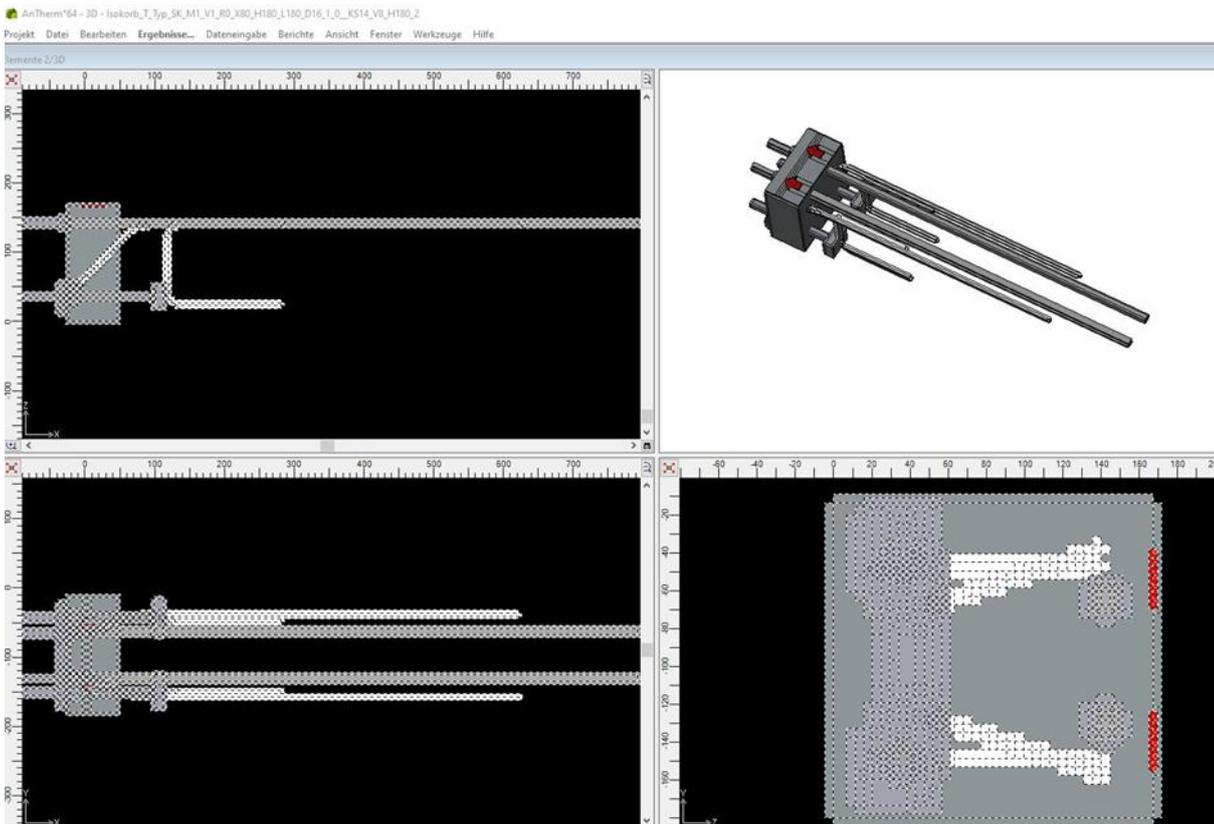
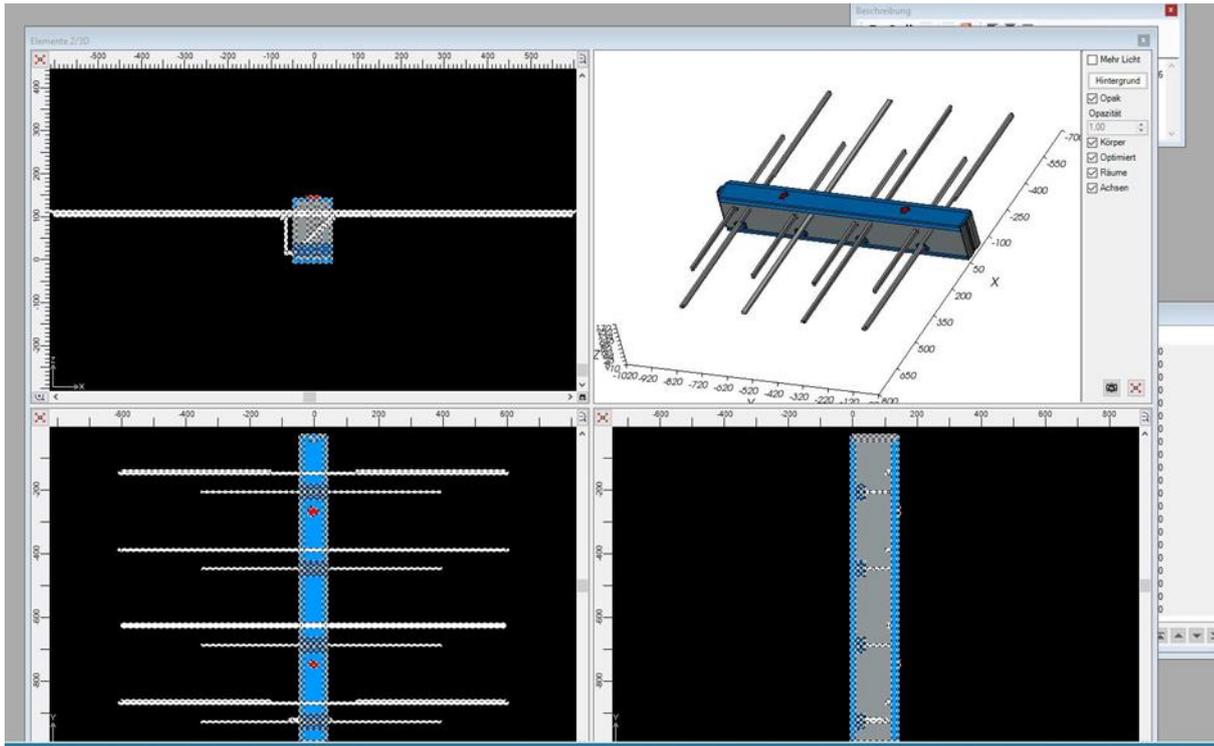
- IFCEXTRUDEDAREASOLID,
- IFCFACETEDBREP,
- IFCFACEBASEDSURFACEMODEL,
- IFCBOOLEANRESULT,
- IFCPOLYGONALBOUNDEDHALFSPACE

utilizing the auxiliary commands

- IFCRECTANGLEPROFILEDEF,
- IFCARBITRARYCLOSEDPROFILEDEF,
- IFCPOLYLINE,
- IFCCIRCLE

and others. If the objects have been assigned materials, AnTherm automatically imports the material names.

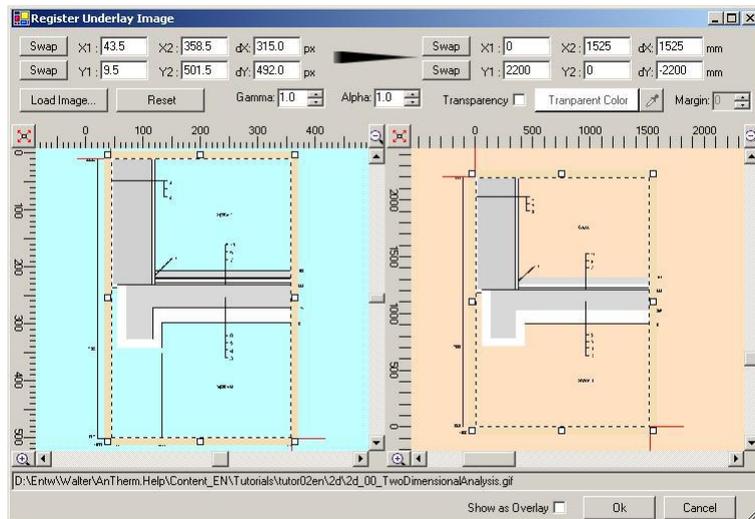




Register Image underlay/overlay (Dialog)

Any arbitrary image file can be loaded within that image registration dialog (button "Load Image") and located properly within the model coordinate system: exact location of two points of the picture will be specified within [workspace coordinate system](#). The image registered by such can be displayed within [Elements 2D window](#) (either as an underlay or as partly transparent overlay) and used for better orientation during the entry of elements during the model creation. **Remark:** The planar underlay is registered at the plane shown within the window. Within the context of [Elements23 window](#) (i.e. a [3D-Project](#)) up to three underlays can be registered within respective [quadrants](#).

This dialog can be reached via the [Main Menu-Import](#) or from the [context menu of the Element 2D window](#). **Remark:** If there is an image defined already (registered) then the image registration data can be adjusted within this dialog.



The left part of the window displays the loaded image within its pixel coordinate system. Entry of pixel coordinates of two points of the image defines the initial part of the transformation to the workspace coordinate system. Input can be done either by numerical entry or by dragging the selection rectangle with the mouse - the selection rectangle will automatically align (snap) to whole pixel coordinate values or to pixel boundary values.

Graphical user interface elements and their behaviour are identical to [Elements 2D window](#).

Within the right part of the window the position of the two chosen image points within the [workspace coordinate system](#) has to be defined (in millimetres). Input can be done either by numerical entry or by dragging the selection rectangle with the mouse.

Important: Please observe, that coordinate values of pixel coordinate system (left image) are ordered from top toward bottom!

The location of chosen points is emphasized in both image views by red cross marks.

In addition to position registration some optimization of the display of the image in [Elements 2D window](#) can be reached by adjusting the transparency colour or colour value range (that colour range will be not rendered and result in transparency effect).

To make the image display brighter the Gamma-Value can be adjusted. For an overly display the transparency of the image can be set with its Alpha-Value.

One can decide if the image will show as an underlay (i.e. below elements and occluded by them) or as overlay (on top of elements). For the latter case it is advisable to choose partly transparent rendering (Alpha < 1.0).

Remark: These detailed settings can be easily adjusted directly from the [context menu of the Element 2D window](#) also.

Important: Data entered here, including the absolute path to the image file, will be saved to the [project file](#). Provided that several projects can reference the same image file as an underlay or overlay - moving the project file to a different folder does not change the image file path. On the other hand, moving or renaming the image file itself will lead to the situation, that it cannot be loaded again and must be re-registered.

Important: The data of the image itself is kept within the image file. Only the path name to the image file is stored within the [project file](#).

Following image file formats can be loaded: JPG, BMP, GIF, TIF, PNG

See also: [Main menu](#), [Coordinate system](#), [Context menu of the Elements 2D window](#), [Project file](#)

Export to CSV file

The matrix of thermal coupling coefficients (conductance matrix) and of power source distribution keys can be [saved as a CSV file](#).

The CSV file can be read by other application (e.g. Excel) for further processing.

Typically the conductance matrix of coupling coefficients (space to space) is exported. If there are power sources within the construction then distribution table (power source to space) is exported also.

Remark: The export operation is executed in current localization of the operating system (the list separator and decimal comma follow the setting of the operating system).

```

csv.csv - Notepad
File Edit Format View Help
"Leitwertmatrix"; "Room; 50"; "Room; 51"; "Room 52"; "Room 53"; "Room 54"; "Room 60"; "
"Room 50"; ; 0,0162851207771624; 0,00653305582212099; 0,00367110372374964; 0,00
"Room 51"; 0,0162866077312319; ; 0,0159179249428953; 0,00675801640336973; 0,003
"Room 52"; 0,00653312105199845; 0,0159168944441141; ; 0,0165161319841276; 0,006
"Room 53"; 0,00367111544916891; 0,00675796311374555; 0,0165160832109682; ; 0,01
"Room 54"; 0,00190730504685556; 0,00325500745579084; 0,0064031607654508; 0,016338224004
"Room 60"; 0,0310933579426317; 0,0126785119114908; 0,00602836460660655; 0,0035164390536
"Room 61"; 0,0127275076696341; 0,0102435619399865; 0,00643027366440056; 0,0041316258948
"Room 62"; 0,00657014575141624; 0,00651398253581024; 0,00522722320506384; 0,00371843658
"Room 63"; 0,00378663315179622; 0,00412709966283829; 0,00378858929594334; 0,00289509346
"Room 64"; 0,00212325245840293; 0,00240408477102851; 0,00233508885523749; 0,00184534242
"Room 70"; 0,0019448889008624; 0,0022348801935049; 0,00222010362646961; 0,0017797024101
"Room 71"; 0,0025672069139479; 0,0030746353862164; 0,00324624645221164; 0,0027031174677
"Room 72"; 0,00235432822057123; 0,00296324898855399; 0,00338590267411903; 0,00298556674
"Room 73"; 0,00193973692764717; 0,00257901607739209; 0,00321796088420045; 0,00304434490
"Room 74"; 0,0012778494700003; 0,00173517948366874; 0,00224001114359938; 0,002184275079
"Room 80"; 0,00180073599205624; 0,00300742749528244; 0,00562341152313261; 0,01144562998
"Room 81"; 0,00235866080182385; 0,00375471402131212; 0,00637206061988574; 0,00924435908
"Room 82"; 0,002270300557303; 0,00338325980258307; 0,00510792485903398; 0,0059692523309
"Room 83"; 0,00183810056959286; 0,00259546406681639; 0,00357662453254302; 0,00372955317
"Room 84"; 0,00123689888609976; 0,00169476125785701; 0,00222132709205096; 0,00219871929
"verteilungsschluesel"; "Room 50"; "Room 51"; "Room 52"; "Room 53"; "Room 54"; "Room 6
"PS 10"; 0,165612491804464; 0,122075374947577; 0,0592054279229958; 0,0344159788835141;
"PS 11"; 0,0815148903388156; 0,0760064021024949; 0,0544372976949996; 0,0362224386843283
"PS 12"; 0,0452352860083497; 0,0476790343988573; 0,0413637882131152; 0,030448347495473;
"PS 13"; 0,0271732362201315; 0,0305252712690222; 0,0292605368820152; 0,0229099906229638
"PS 20"; 0,0231436897344694; 0,0279708720722923; 0,0298611900802456; 0,0249915806813012
"PS 21"; 0,0354415780731821; 0,042740660522763; 0,0445863338774953; 0,0361922608253974;
"PS 22"; 0,0537950965708977; 0,0660319078061737; 0,0657851789969006; 0,0497077900859061
"PS 23"; 0,0780299553435915; 0,10784787156922; 0,097397448592677; 0,0628041442147625; 0
"PS 30"; 0,0447565184530502; 0,0835236323942037; 0,166856723551277; 0,124330696854428;
"PS 31"; 0,0438468703522871; 0,0656005682523857; 0,0923161120372996; 0,0798156852309734
"PS 32"; 0,0346768909235355; 0,0469750535393764; 0,0586659396794006; 0,0525820009901484
"PS 33"; 0,0251531029809711; 0,03268105526496; 0,0389836230419495; 0,0350915991520053;
"PS 40"; 0,0160547645885744; 0,0216598608464403; 0,027650157851972; 0,026646253324337;
"PS 41"; 0,0223656179556854; 0,0312325923542345; 0,0420372350784866; 0,0422380006492421
"PS 42"; 0,0275051207517133; 0,0410343179942069; 0,0612593228984958; 0,0672377568009716
"PS 43"; 0,0289038833944261; 0,0475436530293608; 0,0840515248749342; 0,111934642089591;

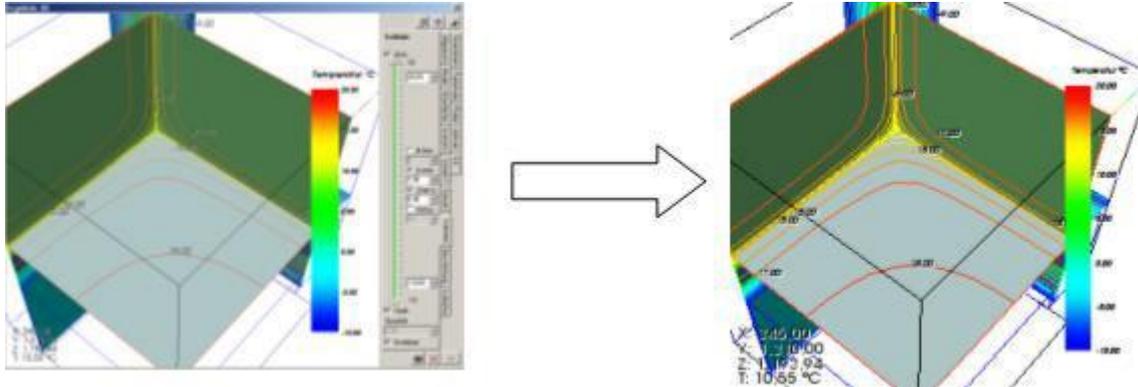
```

See also: Menu [File→Export→Coupling Coeff. Matrix](#)

Export image to file

Graphical images can be [saved to image files \(PNG, TIF, JPG, BMP, GIF\)](#).

The [button „Copy Image“](#) and the menu [„File→Export→Image“](#) (or [„Edit→Copy Image“](#)) provide the transfer function of the picture prepared within a 2D or 3D view to the clipboard and/or to image file.



Important: The 3D image will be [always recreated in specific, defined size](#) and on top of white [background](#) thus being better suitable for printing.

Remark: This function is available if there is 3D graphical window active ([Elements3D, Results3D](#)). Export of 2D views is offered if there is a 2D window active (e.g. [Elements2D](#)).

Remark: Graphical views can be transferred to other applications as [images via the clipboard](#) also.

If the [application setting "clipboard image Save-To-File option"](#) is set to active (true) then you will be offered an option to save (export) the image into an [image file \(PNG, TIF, BMP, GIF, JPG\)](#) - a *file-save-as-dialog* will be presented. Lately chosen export format will be initially offered.

When save operation is repeatedly used the index file name will be offered. If the name recently used ends with digits, then the numeric pattern of same length will be created (Name_0001, _0002 a.s.o.). If the recent name does not end with digits, then a three digits long index will be offered.

Important: The new free index will be used, i.e. if recently Name_0005 has been saved, but yet Name_0001 does not exist, this Name_0001 will be proposed followed by _0002, _0003, _0004 and later _0006,

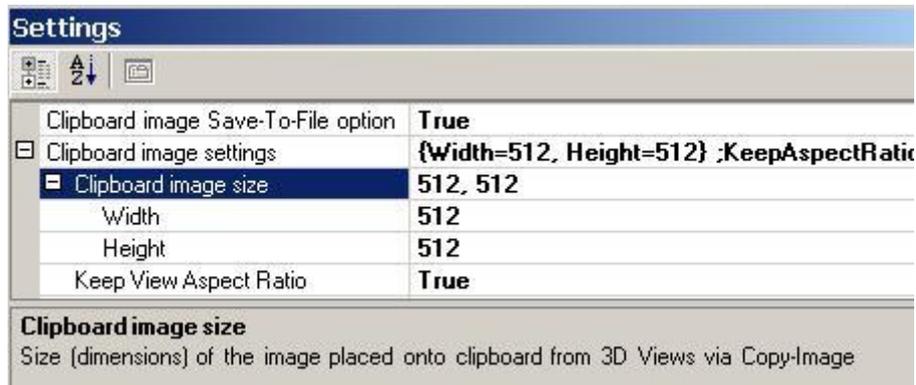
Remark: The option to save images placed onto the clipboard to a file can be set within [application settings](#) (Clipboard image Save-To-File option, [AlwaysAskToSaveGrabbedImage](#)).

Remark: The meaning of the setting "Clipboard image Save-To-File option" can be reversed while the SHIFT-key is held depressed during a click onto the "Photo button". This means, that if the Save-To-File option is active saving the image to the file will not be offered if the Photo-button is depressed during the SHIFT-Key is held depressed. Opposite is also valid: During the Save-To-File option is not active (set to False) and the Photo-button is clicked while holding the SHIFT-key, the dialog offering saving of the image to a file will be presented in contrast.

Dimensions of the image created for 3D views

Important! Dimensions of the image of a 3D view transferred are not equal to the size of the image visible on the screen!

According to the application wide [dimension setting](#) the image will be created at defined [size](#) (within "virtual" window) and transferred thereafter. The [size](#) and proportion ([aspect ratio](#)) of the image specially created for the file storage can be set within [application settings](#).



The input describes the resulting image dimensions in pixels(image points).

Keep View Aspect Ration - application setting

If proportions of the image visible in the window on screen differ from proportions at which the image shall be transferred (i.e. different ratio of width and height) the resulting image transferred will display differently then that on the screen!.

This results from the binding of scaling of the displayed 3D view to the window height:

- try changing the height of the window – 3D object shown will scale accordingly also.
- changing the width of the window does not influence the scaling of 3D objects shown!

This automatic scaling behaviour of the image can be suspended for the image being transferred by the setting „[Keep View Aspect Ratio](#)“ – transferred image will be created at one dimensions actually smaller as set in parameter to show at same aspect ration as the view on the screen.

See also: [Elements 3D](#), [Results 3D](#), [Application settings](#), Menu [File→Export→Image](#), Menu [Edit→Copy Image](#), [Copy image to the clipboard](#), [Keep View Aspect Ratio \(setting\)](#)

Export 3D scene to file

Graphical 3D views can be saved to [saved to 3D scene files \(VRML, IV, OOGL, ...\)](#).

In contrast to two dimensional images which can be [saved to image files](#) or transferred [via clipboard](#), the three dimensional scene file is ideal to be transferred to other three dimensional application for further processing.

A 3D scene consists exclusively of 3D object which can be combined with available building constructions in other applications and rendered therein also (rotated, lighted, over sampled etc.)

A 3D scene file can be offered within a 3D viewer (integrated in modern internet browsers) and being presented on the internet for virtual viewing.

Remark: This function is only available if there is a 3D window ([Elements3D](#), [Results3D](#)) active.

A 3D scene can be saved to the file in one of following formats:

- X3D (.x3d)
- X3D-Binary (.x3d) – default
- POV-Ray (.pov)
- VRML 2.0 (.vml)
- Geomview (.oogl)
- OpenInventor 2.0 (.iv)
- Wavefront ASCII (.obj and .mat)
- Renderman (.rib)

Lately chosen export format will be initially offered.

A confirmation (or an error) message is displayed upon export completion.

When save operation is repeatedly used the index file name will be offered. If the name recently used ends with digits, then the numeric pattern of same length will be created (Name_0001, _0002 a.s.o.). If the recent name does not end with digits, then a three digits long index will be offered.

Important: The new free index will be used, i.e. if recently Name_0005 has been saved, but yet Name_0001 does not exist, this Name_0001 will be proposed followed by _0002, _0003, _0004 and later _0006,

Remark: An export to a Wavefront OBJ file <name.obj> will create the MAT file (contains material properties) <name.mat> also.

Remark: An export to a Renderman RIB file creates on the .rib file only. TIF image is not created. To render the file RenderMan or a RenderMan-Emulator (e.g. Blue Moon Ray Tracer BMRT) is required. This export converts against following Renderman shaders:a) Normal property, no texture map - plastic.sl, b) Normal property with texture map - txtplastic.sl; Both shaders must be compiled into the rendering package used to process the file.

Important: The RIB and OBJ exporter create (also) a rendering based on the size constraints for [Image Exporter \(Width/Height\)](#).

See also: [Export image to file](#), Menu [File→Export→Scene 3D...](#)

Application settings (dialog)

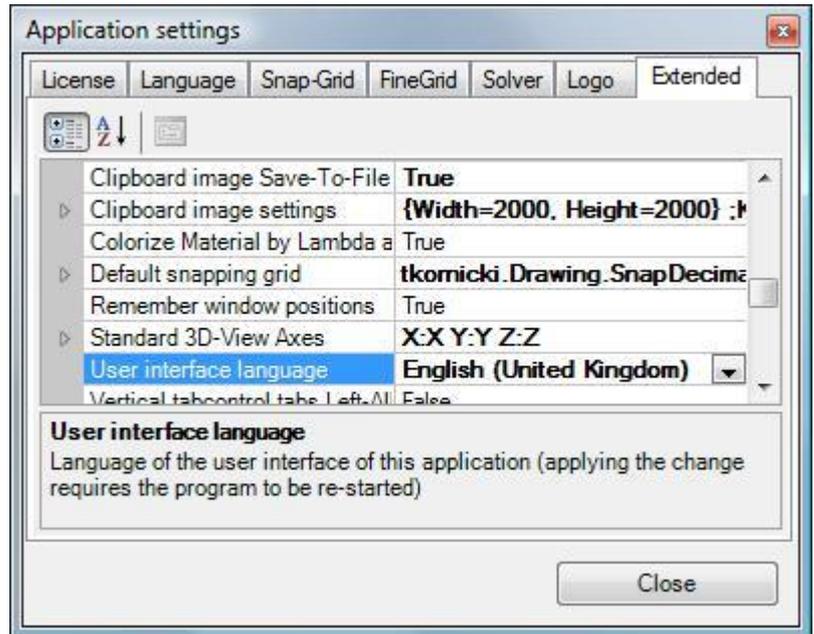
The (extensive) set of application settings can be accessed via the Menu *Edit*→*Settings* from the [main menu](#) and adjusted within the dialog window.

Amongst other following settings are available:

- [language of the user interface](#) (currently English or German)
- [loading the recently edited project](#),
- automatic [calculation of secondary functions](#) (heat flux, relative humidity on surface etc.),
- [standard directory](#) to save project to,
- user defined [logo for report headings](#)
- etc.

Settings shown and adjusted in this window are applied to the application and are independent from project specific parameters (compared to current fine grid parameters of the project, for example, saved with the project).

Changes to some parameters require an application restart (e.g. User interface language) - if so it is emphasized in the description specific parameter.



Parameters which are composed of a set of detailed settings are shown with a plus symbol to the left of the parameter name. A click onto the plus symbol will expand the list of detailed setting offering them for editing.

The bottom part of the window displays the description of the parameter currently selected. The height of this part of the window can be adjusted by dragging the splitter bar.

Every change to the settings are applied immediately (eventually requires application restart too). The button "Close" or ESC-Key close this window.

Evaluation	
Relative Close-Up Error limit	Relative Close-Up Error maximum criteria for coupling coefficient determination. The value set here is used to show precision missed marks (*) and (**) within the coupling coefficient precision information . Remark: Further precision analysis of the calculation shall be considered if current error values are above this limit. Default: 0.0001 (defined by EN ISO 10211:2008); Range: 10 ⁻¹⁰ to 1.0
Relative Close-Up Error - Warn if above	Display a warning message if precision criteria (limit for the relative close-up error of all boundary conditions) is not fulfilled. Default: On
Standard Boundary Conditions	Default values of boundary conditions (temperature and relative air humidity) for the evaluation. These values are used only during the first evaluation of a project. Following defaults are defined: <ul style="list-style-type: none"> • Temperature of the first space (default: -10°C) • Relative humidity of the first space (default: 80 %) • Temperature for all other spaces (default: +20°C) • Relative humidity for all other spaces (default: 53 %) Spaces are sorted alphabetically by their name. All but first space will receive these values by default. First space will be assigned its specific values. Remark: Relative humidity values apply only to evaluations of vapour diffusion. These default values can be set only if DAMPF2/3DIM license option is available. Remark: If the project template "projekt.xml" contains boundary conditions already, that will have the priority over the values specified here.
f_{Rsi,min} design factor for condensation assessment	Minimum value f _{Rsi,min} of the temperature factor of interior surfaces for the condensation-free assessment (within results report). Undercutting this value should mean that condensation free assessment criterion of interior surfaces is not fulfilled. Important: The value f _{Rsi,min} is dependent on interior climate (temperature, air humidity) and can be referred within local regulations.

	Default: 0.69, Range: 0.0-1.0
f_{Rsi,min} design factor for mould-free assessment	Minimum value $f_{Rsi,min}$ of the temperature factor of interior surfaces for the mould-growth-free assessment (within results report). Undercutting this value should mean that mould growth free assessment criterion of interior surfaces is not fulfilled. Important: The value $f_{Rsi,min}$ is dependent on interior climate (temperature, air humidity) and can be referred within local regulations. Default: 0.71, Range: 0.0-1.0
f_{Rsi,min} design factors - Warn if undercut	Display a warning message within results report if mould- or condensation-free criteria is not fulfilled ($f_{Rsi,min}$ design factors are undercut). Default: On
f_{Rsi} - Two-Space only evaluation	Evaluate f_{Rsi} temperature factors (within results report) only if exactly 2 spaces (temperatures) and no further boundary conditions exist in the model. Default: Off (temperature factor are always calculated)
Always Expose Graphical Evaluation	The graphical evaluation window (Results 3D window) will be opened automatically each time boundary conditions are applied . If this setting is turned off (False) the graphical evaluation, if not opened yet, will be not initiated - computationally very intensive and memory consuming evaluation is not requested. Only the results report will be created and exposed. Default: On
Evaluate secondary functions	Secondary functions (like heat flux, relative surface humidity, etc.) will be calculated for graphical evaluation. If this setting is turned off (False) only temperature field will be calculated for Results 3D evaluations. This saves significantly time needed for calculation and reduces memory demand during evaluation. Default: On
Evaluate which secondary functions	Allows selective selection of secondary function to be calculated for graphical evaluation. This provides the ability to evaluate only needed functions by simultaneously reducing the memory demand and calculation time otherwise spent for functions the user is not interested in (surface humidity, heat flux, heat stream vector field, saturation pressure etc.). <ul style="list-style-type: none"> • Temperature °C • Heat flux W/m² (magnitude of the vector) • Vector field of heat stream W/m² • Relative condensing surface humidity % (only at component's surface), respectively: <ul style="list-style-type: none"> • Condensation: "Condens. RH %" - air humidity limit leading to surface condensation [0-100%] • Mould growth: "Mould RH %" - air humidity limit leading to mould growth on surfaces (i.e. 0.8*Condens.RH) [0-80%] • Corrosion: "Corrosion RH %" - air humidity limit leading to corrosion of metals on surfaces (i.e. 0.6*Condens.RH) [0-60%] Further secondary function are available with the active VAPOUR-Option: <ul style="list-style-type: none"> • saturation pressure hPa • partial pressure hPa • vapour pressure difference hPa (= saturation - partial pressure) • relative core humidity % (relative interstitial humidity; interstitial vapour saturation ratio = partial pressure/saturation pressure * 100) • vapour diffusion flux mg/m²h • vector field of vapour diffusion stream mg/m²h  <p>Remark: Application settings specific to the VAPOUR-Option (partial pressure, pressure difference, interstitial core humidity, vapour flux and vector) can only be changed if the DAMPF2/3DIM license option is available (otherwise are shown as Read-Only). Remark: This setting is only meaningful if the setting "Evaluate secondary functions" is turned on. See also: Parameter setting "Evaluate secondary functions", General (control panel) Default: By default all secondary functions are evaluated. It is advised to turn off evaluation of not used secondary functions to reduce the computational time and memory demands.</p>
Restore last Result3D Parameters	Defines how (and if) to restore the Results3D Parameters on first display of the Results3D window (the combination of evaluation parameters is saved when the Results3D window is closed). <ul style="list-style-type: none"> • No - parameters will not be restored; predefined default combination of parameters will be used. • Restore - exact combination of parameters will be restored; some restored numerical values might not be suitable for the model currently evaluated. • RestoreAdapt - combination of parameters will be restored and some numerical parameters will be adjusted to reflect currently evaluated model (e.g. Tube Radius etc). Default: RestoreAdapt (<i>earlier versions, up to 4.80: No</i>)

User Interface	
Autosynchronize materials & surfaces	Defines if the project specific material list and surface list are automatically synchronised (on each element selection change). This can be also change on the context menu of both lists ("Autosynch. from Elements") See also: Material window , Surfaces window Default: On
Colorize material by Lambda automatically	Defines if the material colour is automatically calculated and changed then the value of Lambda changes (by LambdaToColor-Table). When off the colour assigned to the material will be retained during changes to the value of Lambda. See also: Material Editor window , Element Editor , Colour setting file LambdaToColor.ColorList Default: On
Clipboard image settings / Clipboard image size	Dimensions (in pixels) of images shot from 3D views and placed onto the clipboard (via Copy Image). Please keep proportions (width/height) of the image targeted for the clipboard same to proportions of the view on the screen. If proportions differ the resulting image placed on to the clipboard will probably differ (in its scaling) from the view seen on the screen. See also: Keep View Aspect Ratio , Copy image to clipboard , Export image to file Default: 2000x2000 (Range: 2-8192 or less after Bitmap-Test)
Clipboard image settings / Keep View Aspect Ratio	If turned on proportions of the image placed on the clipboard reflect these of the image shown on screen. Preset dimensions (see above) of the image will be treated as maximum values for the resulting image (the resulting image will fit into the rectangle given by the dimensions specified). See also: Clipboard image size , Copy image to clipboard , Export image to file Default: On
Clipboard image Save-To-File option	Defines if after copying the image onto the clipboard an option to save the image to the file (Save Image dialog) shall be offered also. Default: Off
Remember window positions	Positions of windows on the screen shall be saved and windows shall reopen at positions saved lately. Default: On
Background 2D colour	Background colour of 2D views (Elements 2D , Elements 2/3D , ...) Default: Black
Background 3D colour	Background colour of 3D views (Elements 3D , Results 3D , ...) Default: WhiteSmoke
User Interface Language	By default the user interface of the application reflects the language of the operating system (standard/default language). The user can change this setting! If the language setting is not supported by the application (current distribution provides only English and German), the application will fall back to English. Be aware of: Country specific localisation Default: Language of the operating system (if available, otherwise default will be English)
Standard 3D-View Axes	Defines titles displayed along axes within 3D view (used in Elements 3D and Results 3D window) Default axes titles: X, Y, Z
Default snapping grid	Defines settings of Snap-To-Grid functions (used in 2D Element Editor) See also: Snap-To-Grid parameter Default: All snapping On, X(0,100) Y(0,100) Z(0,100), Distance=10
Vertical tabcontrol tabs Left-Alligned	Tabs of the control panel within Results3D window will show at the left edge of the window (instead of the default at right side). This setting allows to change the placement of tabs to the left side of the window which might be common to architects and engineers. This reflects familiar tilt of the head to the left to read tab labels. Default: Off
Vertical tabcontrol tabs with Labels	Tabs of the control panel within Results3D window will show labelled with Text Default: On
Vertical tabcontrol tabs with Icons	Tabs of the control panel within Results3D window will show graphical symbols also. Default: On
Project Files	
MRU Project Count	The Number of recently processed project names (paths) the application shall remember. (value range 0-200, -35 with menu shortcuts) Default: 8 See also: Main menu
Reopen last project	The last project saved or opened prior to application exit will be reopened automatically on next application start. Default: On
Overwrite project without asking	The project file will be overwritten without any warning or approval (only if the name for the project file is already set). If the name of the project file is already set (either saved or loaded) saving the project is executed without any additional warnings. This simplifies the step „ Results... “ – when combined with

	AlwaysAskForFineGridParams =FALSE and AlwaysAskForSolverParams =FALSE the application shows results directly. Default: On
Always Ask To Save Project	You will be reminded to save current project prior to any destructive operations (this the default behaviour). Prior to disposing data of current project (New , Load , Convert etc.) the user will be asked to save the project. Answering "No" will continue with the operation requested without saving the project (which potentially result in losing current project data). Answering "Abort" will immediately cancel the requested operation. Default: On
Projects	
Create Directory For Reports	When creating a new project, create a directory where the reports and related images are stored. Default: Off
Grid	
Always ask for fine grid parameters	User will be asked to adjust and confirm fine grid parameters (the dialog Fine-Grid parameter will be shown) on each computation. Default: On
Default fine grid parameters	Default values of parameters used by fine grid generator applied to new projects See also: Fine-Grid parameter
Solver	
Automatically close Solver Form	The Solver window will close automatically when the computation is finished. If this setting is turned off (False) the application will wait until the user manually closes this window (by pressing the button "Close"). Keeping the window visible can be helpful when further detailed analysis of messages produced by the solver is required (e.g. at some error conditions). Default: On
Animate Progress on Solver Form	Show the progress animation on Solver window . If this setting is turned off (False) the animation will not show thus saving CPU time. Default: On
Always ask for Solver parameter	User will be asked to adjust and confirm Solver parameters on each computation (the dialog Solver parameter will be shown). Default: On
Default Solver parameters	Default values of parameters used by the simulation engine applied to new projects. See also: Solver-Parameter
Run Solver On Multiple Processors	Run the Solver utilizing multiple processors or processor kernels. The maximum number of processors utilized is set with the parameter " Max Number Of Solver Processors ". Remark: This setting can be changed only if there are multiple processors (or processor cores) available in the system used. Remark: This setting requires a valid license feature MULTICORE . See also: application setting "Option MULTICORE active" . Default: On (subject to license feature MULTICORE and the OS reported number of processor above 2). The user shall adjust the Number Of Solver Processors to achieve optimum performance.
Max Number Of Solver Processors	The Maximum Number Of Processors utilized by Solver (if it shall run on multiple processors). The number entered is limited to the actual number of processors (or processor cores) currently installed and available in the system. The user shall adjust this number to achieve optimum performance . 0 (default) - utilize all available processors/cores. Remark: This setting can be changed only if there are multiple processors (or processor cores) available in the system used. Remark: This setting requires a valid license feature MULTICORE . See also: application setting "Option MULTICORE active" .
Warn on excessive Equation Size	Display a warning message in the Solver if the equation size is above the value specified (0 - disable warning). Default: 1000000
Directories & Files	
Material Database	Path to the Material database Default: <installation folder of the program>\Baustoffe.xml
Simulation Folder Cleanup (Reminder)	Set the number of days after the latest invocation of the Simulation Folder Cleaner Tool to remind you about the cleanup of intermediary simulation results eventually needed. If you ignore the reminder when shown, the setting is raised by 1 automatically. Default: 30 (days) 0 - no reminder.

User specific Logo	<p>Path to the User specific logo file.</p> <p>The logo image is displayed in the heading of reports, as middle item of the heading, placed left to the Copyright-Info, 6cm wide and 1.5cm high. To avoid distortion of the picture keep the width to height ration of 4:1 (for example 800x200 pixels).</p> <p>If no logo image shall be shown on reports leaving this field empty is allowed.</p> <p>Default: <installation folder of the program>\CustomLogoBitmap.bmp</p>
Default project directory	<p>The Path to the folder to which project files shall be saved. This setting is only used for dialogs of save/load menus if the project currently processed has not been saved to file ever yet (this is valid for new projects).</p> <p>Default: "My Documents\AnTherm"</p>
öbox data download folder	<p>Path to the store of öbox-data which then are shown within material database window.</p> <p>Default: <LocalApplicationData>\OeBOX_Downloaded, typical: C:\Documents and Settings\<USER>\Local Settings\Application Data\OeBOX_Downloaded</p>
baubook(öbox) data included	<p>Include baubook(öbox)-data (if available) within material database window. This, if turned off, allows you to exclude baubook(öbox)-data from the view even if the baubook(öbox) data file is available. You will have to close and reopen the material database window to fully apply change in setting of this switch.</p> <p>Default: On</p>
BPHDB.COM data included	<p>Include BPHDB.COM data (if available) within material database window. This, if turned off, allows you to exclude BPHDB.COM data from the view even if the BPHDB.COM data file is available. You will have to close and reopen the material database window to fully apply change in setting of this switch.</p> <p>Default: On</p>

Special (Advanced) settings

Settings collectively grouped here are used to adjust behaviours and techniques used by the program. Such special adjustments must be accounted also to reflect various differences in hardware used, which cannot be automatically analysed and might lead to undesirable behaviour of the application.

Advanced UI Settings	
Alpha (surface transmittance) hidden	Hide the Alpha field (surface transmittance $\alpha=1/R_s$) within space surface property input. When turned off both reciprocal inputs of surface resistance R_s and transmittance α are offered (compatible with versions prior to 6.102) on Element Editor and Surface Editor forms and reported on Modelling Report . Default: On
Elements23Form Panel Contents	Contents shown within quadrants of Elements23 window can be adjusted and contain choices of parallel projections XY, XZ, YX, YZ, ZX, ZY, the 3D view and element selection list . One can also do without showing anything within quadrant (in most cases the 3D view might be obsolete). Default: <ul style="list-style-type: none"> • Left-Bottom: XY • Left-Top: XZ • Right-Bottom: ZY • Right-Top: 3D
Primary Edit Windows 2D-Project	Defines which Editing windows will automatically open for 2D project . Default: Description , Element Editor , Element selection , Elements 2D
Primary Edit Windows 3D-Project	Defines which Editing windows will automatically open for 3D project . Default: Description , Element Editor , Element selection , Elements 23
Primary Edit Windows 3D-Layered-Project	Defines which Editing windows will automatically open for layered 3D project . Default: Description , Element Editor , Element selection , Elements 2D , 3D-Layers , Elements 3D
Restore last Camera Position	Restore to the recent Camera position on initial display of a 3D window (applies to Elements3D , Elements23 , Results3D). <ul style="list-style-type: none"> • No - position will not be restored, only Fit (center, scale); predefined default position will be used. • Restore - exact position of the camera will be restored; might not be suitable for the model currently evaluated. • RestoreAndFit - position of the camera will be restored followed by a Fit (center, scale) Default: RestoreAndFit
Show using Visual Styles	When enabled Buttons and other controls will render with (new) visual style, e.g. rounded. Default: On You will have to end and restart the application to consistently apply the change of this setting. <u>Remark:</u> For Windows XP the default is "Off" to overcome some compatibility issues resulting from implementation error on that systems when showing text on vertical tabs of the tab control which are not rendered properly if visual styles are enabled. The user can still enable that setting manually.
Help Window shown TopMost	Help window will be shown topmost to application windows and automatically minimize with the application (this is the windows default behaviour). When turned off (the default) the help window can be set behind the application window via typical desktop window manager. <u>Remark:</u> It applies only to Help.ShowHelp but not the HelpProvider (invoked via F1 key) thus the effective behaviour depends on how the help window was opened. If HelpProvider opens the Help window (user pressed F1 key) then the default (TopMost) behaviour will be always retained. If Help.ShowHelp opens the Help window (via Help-Contents, Tutorials, etc.) then the behaviour set here will be retained also for subsequent F1 requests. Default: Off
Advanced Settings	
Alternative License File AnTherm.HID	Alternative path to the store of the license file AnTherm.HID . Important: This setting, if not empty, takes precedence over the " Alternative list of license files AnTherm.HID ". If left empty (default) the license file AnTherm.HID is looked for within the installation folder, and, if not found there, within the subfolder DemoLicense of the installation folder.

	<p>Important: A valid license file MUST be named AnTherm.HID. AnTherm will create a temporary local copy of the file provided to also allow network stores without additional performance penalties.</p> <p>ZIP Archive: specifying a ZIP archive storing the license file AnTherm.HID in its top level directory is also supported. If the file specified does not exist, this setting is simply ignored. Default: empty</p> <p>Important: You will have to end and restart the application to consistently apply the change of this setting.</p>
Alternative List of license files AnTherm.HID	<p>The list of alternative paths to multiple stores of license files AnTherm.HID of which the first valid or first marked expired will be employed.</p> <p>Important: The setting "Alternative license file AnTherm.HID", if not empty, takes precedence over this one (which then will be ignored). If left empty (default) the license file AnTherm.HID is looked for within the installation folder, and, if not found there, within the subfolder DemoLicense of the installation folder.</p> <p>Important: A valid license file MUST be named AnTherm.HID. The name of a ZIP archive storing the file AnTherm.HID can be set alternatively too. AnTherm will create a temporary local copy of the file provided to also allow network stores without additional performance penalties. If the file specified does not exist, this setting is simply ignored. Default: empty</p> <p>Important: You will have to end and restart the application to consistently apply the change of this setting.</p> <p><u>Remark:</u> The purpose of this setting is, in the case you have multiple license files bound to different hardware identifiers (e.g. dongles), eventually with varying functional scope. Switching the dongle the first license file matching it will be validated.</p>
Asynchronous Evaluation Builds	<p>Turn this setting off to force the application to execute all its parts sequentially (otherwise executing in parallel and possibly utilizing multiple CPUs if available) on weaker computer or for testing purposes. Default: On</p>
Windows mode MDI	<p>Show windows in MDI mode (you will have to restart the application to apply this setting). MDI: All application windows will be shown as sub-windows within the main application window.. MDI mode is especially useful with lower screen resolutions (<1600x1200). The MDI main window is scrollable. The main MDI window cannot be sized below 600x400 pixels (such small windows size is much too small for efficient application use, but might be convenient for the first application start).</p> <p>SDI: While in non-MDI mode (WindowModeMDI=false, also called "SDI mode") the height the main form will be reduced to lowest possible size to show the main application menu only. The main window will be declared as "Top-Most" ensuring its visibility on top of all other application windows to avoid hiding of the main application menu. Default: On</p>
Fade Info-Dialog Gradually	<p>The About dialog shall appear by decreasing its opacity in small steps. On some (older) machines this might result in undesirable flickering effect, thus this setting can be turned off if such behaviour is observed. Default: On</p>
Multiple app-instances allowed	<p>Simultaneous execution of multiple instance of this application can be allowed by turning this setting to on. This might be useful when components or their parts shall be quickly transferred between projects via clipboard (copy/paste) without the need to close the active project and open another one in between. Default: On (was "Off" till V.5.88).</p> <p>Warning: Simultaneous execution of multiple instance of this application might result in race conditions between instances and lead to severely erroneous results. You shall never edit same project originating from/in the same directory simultaneously or otherwise the application will behave unexpectedly.</p>
Option VAPOUR active	<p>The VAPOUR option can be switched off by the user: Within the MainForm the application setting ComputeVaporTransferSolution will be reset The EditControl of ElementEditor will not show the Mu value The MaterialEditorControl of the MaterialsList will not show the Mu value Within the SolutionFactory the setting ComputeVaporTransferSolution will be reset The SolverParamsControl will disable the setting of ComputeVaporTransferSolution <u>Remark:</u> This setting requires a valid license feature DAMPF2/3DIM.</p> 

Option HARMONIC active	<p>The HARMONIC option can be turned off by the user.</p> <ul style="list-style-type: none"> • Within the MainForm the application setting ComputeTransientSolution will be reset • The EditControl of ElementEditor will not show the Rho and Ce values • The MaterialEditorControl of the MaterialsList will not show the Rho and Ce values • Within the SolutionFactory the setting ComputeTransientSolution will be reset • The SolverParamsControl will disable the setting of ComputeTransientSolution <p><u>Remark:</u> This setting requires a valid license feature HARMONIC. <u>Remark:</u> Suspending HARMONIC option turns off the TRANSIENT option too.</p>
Option TRANSIENT (incl. HARMONIC) active	<p>The TRANSIENT option can be turned off by the user - when time dependant evaluations are not needed (i.e. no entry of Periodic Boundary Conditions, Evaluation Time Point and no Timeline- nor Animate-Evaluations)</p> <ul style="list-style-type: none"> • The SolverParamsControl will limit the number of harmonics to 6. • The MainForm will suppress time dependant evaluations <p><u>Remark:</u> This setting requires a valid license feature HARMONIC/TRANSIENT. <u>Remark:</u> Turning the TRANSIENT option ON turns on the HARMONIC option too.</p>
Option MULTICORE active	<p>The MULTICORE option can be switched off by the user. When deactivated the optimized execution of calculation (Solver) and many further evaluation components of the application will be suppressed.</p> <ul style="list-style-type: none"> • Solving the equation system(s) - the Solver • Loading the solution from files in solution folder • Combining solution vectors at specific boundary conditions • Fourier analysis and synthesis of periodic boundary conditions • Fourier synthesis of time dependant solutions • Creation of results data for 3D visualization (primary and secondary functions), timelines and animation <p><u>Remark:</u> This setting requires a valid license feature MULTICORE See also: application settings "Run Solver On Multiple Processors", "Max Number Of Solver Processors".</p>
Option STEREO3DVIEW active	<p>The STEREO3DVIEW option can be switched off by the user. When deactivated stereo rendering capability of 3D windows will not be available. You will have to close and reopen all 3D windows to fully apply change in setting of this switch.</p> <p><u>Remark:</u> This setting requires a valid license feature STEREO3DVIEW</p>
Lately used Settings File	<p>The path to the settings file loaded lately (read only, for information only).</p>
OpenGL Settings	<p>Control parameter for the OpenGL system library <u>Remark:</u> Changes to settings in this group require all 3D-windows to be reopened or the application to be restarted to apply them.</p>
ResolveCoincidentTopology ToPolygonOffset	<p>This setting ensures enhanced visualization of lines drawn along surfaces. When turned off lines (e.g. isolines, edges etc.) will be drawn coplanar to surfaces, possibly leading to undesired visual effects (e.g. lines only partly visible, partly obscured). On some weaker graphics systems (e.g. DELL Latitude D505 with Intel Extreme Graphics 855GM) this setting might be turned off or otherwise rendering of surfaces is not correct. Default: On</p>
UseBackingStore	<p>Turn on/off using backing store. This may cause the re-rendering time to be slightly slower when the view changes. But it is much faster when the image has not changed, such as during an expose event. Default: On <u>Remark:</u> On some systems it is undesirable to turn this option on - areas of covered partly by other windows might not be redrawn correctly during normal rendering.</p>
UseImmediateMode Rendering	<p>Turn on/off flag to control whether data is rendered using immediate mode or not. Immediate mode rendering tends to be slower but it can handle larger datasets. If you are having problems rendering a large dataset you might want to consider using immediate mode rendering. Default: Off (i.e. RetainedMode-Rendering)</p>
ExplicitBackfaceProperty	<p>Will assign BackfaceProperty=FrontFace for modelSliceX, modelSliceY, modelSliceZ, modelSliceZ, sliceY, sliceX, isosurface, surface; It is used to resolve rendering issues on (very few) systems rendering only one side of surfaces. (See workaround in FAQ too). Default: Off (i.e. default behaviour)</p>
DesiredFlyToSeconds	<p>Define the time used for animating to the particular isometry applied. Default: 2 seconds (Range: 0 - 100).</p>

	Remark: Actual number of frames shown during the animation results automatically from current rendering time. Rendering time depends highly on the object complexity and its current transform and varies during the animation - faster 3D graphics systems will show more animated frames thus the resulting move will look natural.
Interactor DesiredUpdateRate	Desired number of frames per second while the 3D view is rotated or zoomed with the mouse interactor . Default: 2 fps (Range: 0.0001-10000). It shall always be larger then the StillUpdateRate Remark: Setting InteractorDesiredUpdateRate to 0.0001 will practically turn the LOD functionality off (full detailed image on each rendering).
Interactor StillUpdateRate	Number of frames per second while the 3D view has became still after being rotated or zoomed with the mouse interactor . Default: 0.0001 fps (Range: 0.0001-10000). It shall always be smaller then the DesiredUpdateRate.
TubeNumberOfSides	Number of sides of the Tube representation (Streamlines , Isolines). Default: 4 (Range: 3-16) Remark: After changing this setting you will eventually have to restart the visualization.
ConeResolution	Number of facets used to represent the cone (as used by Vectors/HedgeHog). =0: a line; =1: a single triangle; =2, two crossed triangles; >2, a 3D cone (with resolution number of sides). Default: 4 (Range: 1-16) Remark: After changing this setting you will eventually have to restart the visualization.
ConeRadiusToLengthRatio	The radius of the cone base as ratio of its length (as used by Vectors/HedgeHog). 0.15 ~three times long as wide; 0.25 twice longer as wide. Default: 0.15 (Range: 0.001-0.5) Remark: After changing this setting you will eventually have to restart the visualization.
ArrowResolution	Number of facets used to represent the arrow shaft and its tip (as used by Vectors/HedgeHog). =1: a single triangle and single rectangle; =2: two crossed triangles and single rectangle; >2: a 3D tip cone and 3D shaft (with resolution number of sides). Default: 4 (Range: 1-16) Remark: After changing this setting you will eventually have to restart the visualization.
ArrowTipRadiusToLengthRatio	The radius of the arrow's tip cone base as ratio of arrow length (as used by Vectors/HedgeHog). Default: 0.15 (Range: 0.001-0.5) Remark: After changing this setting you will eventually have to restart the visualization.
ArrowShaftRadiusToLengthRatio	The radius of the arrow's shaft as ratio of arrow length (as used by Vectors/HedgeHog). Default: 0.05 (Range: 0.001-0.5) Remark: After changing this setting you will eventually have to restart the visualization.
ArrowTipLengthToLengthRatio	The length of the arrow's tip cone as ratio of arrow length (as used by Vectors/HedgeHog). Default: 0.4 (Range: 0.001-0.999) Remark: After changing this setting you will eventually have to restart the visualization.
StereoAnaglyphColorSaturation	The anaglyph color saturation factor (as used by anaglyph stereo rendering mode). 0.0 means that no color from the original object is maintained, 1.0 means all of the color is maintained. The default value is 0.65. Too much saturation can produce uncomfortable 3D viewing because anaglyphs also use color to encode 3D. Default: 0.65 (Range: 0.0-1.0).
StereoAnaglyphColorMask	The anaglyph color mask values are two numbers which are bits mask that control which color channels of the original stereo images are used to produce the final anaglyph image (as used by anaglyph stereo rendering mode). The first value is the color mask for the left view, the second the mask for the right view. If a bit in the mask is on for a particular color for a view, that color is passed on to the final view; if it is not set, that channel for that view is ignored. The bits are arranged as r, g, and b, so r = 4, g = 2, and b = 1. By default, the first value (the left view) is set to 4, and the second value is set to 3. That means that the red output channel comes from the left view, and the green and blue values come from the right view. Default: Red_Cyan = R_GB Range: R_GB=Red_Cyan; GB_R=Cyan_Red; G_RB=Green_Magenta; RB_G=Magenta_Green; B_RG=Blue_Yellow; RG_B=Yellow_Blue
Display ErrorAndWarning Window	Warning and error messages of visualization components will be displayed (a dedicated window will be exposed). Default: Off

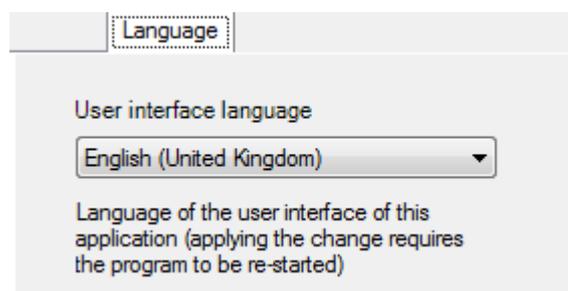
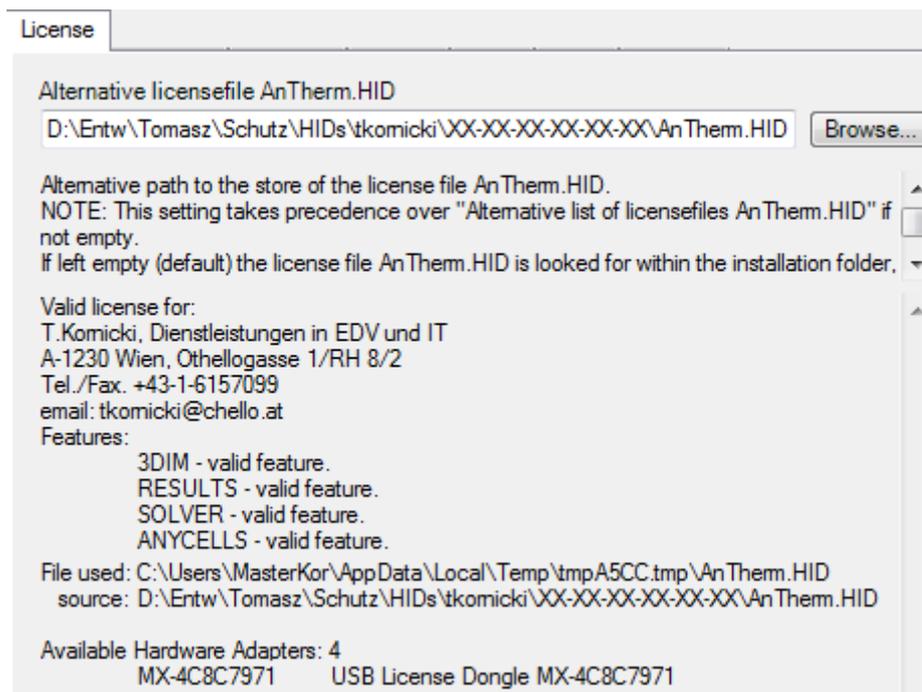
Advanced Solver Settings	
Compute Vapour Transfer Solution inquiry	Always ask if Solver shall create the solution for the vapour transfer problem provided the sufficient data is available but this Computation has not been requested yet. This setting is subject to additional licence restrictions if such apply. Default: On
Compute Transient Solution inquiry	Always ask if Solver shall create the solution for the transient periodic, harmonic, instationary problem provided the sufficient data is available but this Computation has not been requested yet. This setting is subject to additional licence restrictions if such apply. Default: On

Hint: To restore the value of any respective parameter to its default use its context menu (right mouse button) and select "Reset".

Remark: Application wide settings are stored to the [file AnTherm.exe.Settings.xml](#) which is kept within local settings folder of any user.

The lately active tab of the settings window will be save with the application settings and later restored (LastSelectedApplicationSettingsFormTabPageName).

See also: [Settings file AnTherm.exe.Settings.xml](#), [Fine Grid parameter](#), [Solver parameter](#), [Snap-To-Grid parameter](#)



Snap-Grid

Turn Snapping On

Snap to Grid

Step :	- X -	mm	- Y -	mm	- Z -	mm
	100		100		100	
Offset :	0	mm	0	mm	0	mm

Snap to existing Points

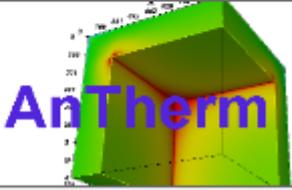
Distance : 100 mm

Logo

User specific Logo

D:\Entw\Walter\AnTherm\bin\Debug\CustomLogoBitmap.bmp

Path to the user specific logo picture. The picture will be exposed in the heading of all reports.
The proportions width/height of the pictures shall be 4/1 to avoid disortion of the picture shown.



You can Drag & Drop a file onto the surface of this control t

FineGrid

Always Ask for Fine Grid parameters

Start Fine Grid at:

Space Surfaces Adiabatic Cut-Off Planes

Power Source Boundaries

Material Changes when λ , μ , $\rho \cdot c$ ratio above:

Step by:

Start Step : mm

Max Step : mm

Step Growth Factor : mm

Suppress fine grid :

all axis directions

direction of X axis Y axis Z axis

Target Model Size:

Target for Model Size # Cells:

Solver

Always Ask for Solver parameters

General Iteration Omega Advanced

Control parameters of solution iterations:

Termination condition (Delta) :

Maximum number of iterations :

Control parameters of vapour transfer calculation:

Compute Vapor Transfer Solution :

Control parameters of transient, periodic, harmonic calculation:

Compute Transient Periodic Solution :

Year (365 days) Day (24 hours) Custom s

harmonics harmonics harmonics

Force Recalculation of Solution :

Settings file AnTherm.exe.Settings.xml

Global [application settings](#) are saved to the file AnTherm.exe.Settings.xml located in the user specific settings folder.

During application shutdown current window positions will also be saved to this file. On next application use these windows will be shown at their last position. If this stored setting is not sufficient anymore and you wish to reset the layout of all application windows, please use the menu function [Windows→Reset windows positions](#). On next application start following this reset all windows will be shown at positions defined in the initial template [AnTherm.exe.LayoutPersistence.xml](#).

Actual filter settings defined in the [Material database window](#) are also persisted.

The settings file also stores latest arrangement of 3D views (the actual isometrics of 3D representation). Given by the view vector and perspective it will be restored when a 3D window is opened again.

Furthermore, this settings file keeps track of all global (standard) evaluation settings lists, isometry lists a.s.o. (these are managed from the "standard" panel of the [isometries window](#) and [results 3D parameters window](#)).

The [list of all folders which appear used by AnTherm as containers for AnTherm projects and simulation results folders](#) is stored in that file also.

Note: The folder used for local settings is located, for example in Windows XP Professional, under following path name:

```
C:\Documents and Settings\\Local Settings\Application
Data\tkornicki\AnTherm\
```

Actual location of this folder depends on the settings of operating system, its version and language choice. The exact path can be found in the About window (menu Help→About...) by looking for the value of the entry:

```
LocalUserAppDataPath:
```

See also: [Application settings](#)

Theoretical background

The calculation of temperature field and heat streams is founded on top of the general concept of base solutions and thermal coupling coefficients.

The theoretical foundation for these concepts for steady state (stationary) calculations can be found within the book „[Wärmebrücken \(Thermal heat Bridges\)](#)“ from W. Heindl, et al. [1] and for the dynamic (transient harmonic, periodic) calculation within writings of K. Kreč [2].

Main feature of theoretical concepts used here has been directly adopted within the "Thermal Bridges Standard" EN ISO 10211-1 [3], hence the stringent [standard's conformance](#) of the program AnTherm is easily provided.

It is noteworthy to reference [the distinctive method of direct and exact determination of temperature factors fR_{si}](#) which applies to multidimensional (three dimensional) cases also.

The numerical methods used in this program build on top of standard mathematics bibliography [4].

The special subjects and methods of optimal over-relaxation are further described in [5].

See also: [Introduction, Why multidimensional calculation?](#), [European standards on thermal bridges](#), [The over relaxation, Primary Concepts](#)

Articles and essays

- [On condensing humidity \(dew point\)](#)
- [Calculating equivalent thermal transfer coefficient of air cavities](#)
- [On calculation of \$\Psi\$ -values \(Psi\) for a 3 space case: Special consideration: Unheated spaces](#)
- [On calculation of \$\Psi\$ -values for building constructions in connection with ground](#)
- [The air temperature in the unheated basement, a case study](#)
- [Heat losses from large buildings through the ground](#)
- [Methods of thermal heat bridge survey in international perspective](#)
- [On three dimensional simulation of thermal behaviour of buildings](#)
- [The building envelope as thermal heat bridge](#)
- [On the Storage of Heat in Building Components](#)
- [Transient, multidimensional calculations by an example of components in contact with soil](#)
- [Heat Transfer in Building Constructions in consideration of Heat Sources](#)
- [A practical method for the assessment of heat bridges with a view to surface condensation](#)
- [Multidimensional vapour diffusion processes](#) 
- [On visualization of heat and vapour diffusion stream in three dimensional models](#)
- [Temperature factor fR_{si}](#)
- [Appropriate responses to the complex problem of finding the cause for mould growth?](#) [NEW]
- [Das Leitwert-Konzept](#) [NEW]
- [Anmerkungen zur EN ISO 13786](#) [NEW]
- [Berechnung der spezifischen Wärmekapazität der Ersatzkonstruktion](#) [NEW]
- [Die wirksame Wärmespeicherkapazität](#) [NEW]
- [Wärmeleitung und Wärmespeicherung in plattenförmigen Bauteilen](#) [NEW]
- [Zur Definition der wirksamen Wärmespeicherkapazität](#) [NEW]

Bibliography

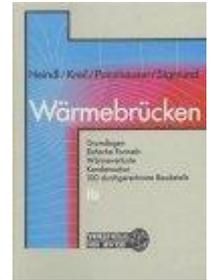
- [1] [W. Heindl, K. Kreč, E. Panzhauser & A. Sigmund, "Wärmebrücken", Springer-Verlag \(1987\) ISBN:3211820248, ISBN:0387820248](#)
- [2] K. Kreč, "Zur rechnerischen Erfassung des Einflusses von Wärmebrücken auf das thermische Verhalten von Gebäuden", Habilitationsschrift, TU Wien (2000)
- [3] EN ISO 10211-1: „Wärmebrücken im Hochbau – Wärmeströme und Oberflächentemperaturen – Teil 1: Allgemeine Berechnungsverfahren". (1996)
EN ISO 10211 "Thermal bridges in building construction — Heat flows and surface temperatures — Detailed calculations". (2007)
- [4] Stoer-Bulirsch; "Einführung in die Numerische Mathematik II"; Springer Verlag Berlin- Heidelberg-New York (1973)
- [5] Björck, Dahlquist "Numerical Methods"; Prentice-Hall, (1983) ISBN:8301042761
- [6] [Raul H. Mendez „Visualization in Supercomputing“, Springer Verlag \(1990\) ISBN:0387971491, ISBN:3540971491](#)
- [7] [K.W.Brodie, L.A.Carpenter, R.A.Earnshaw, J.R.Gallop, R.J.Hubbold, A.M.Mumford, C.D.Osland, P.Quarendon „Scientific Visualization, Techniques and Applications“, Springer Verlag \(1991\) ISBN:3540545654, ISBN:0387545654](#)
- [8] [R.M.Fridhoff, W.Benzon „the second computer revolution – VISUALIZATION“, Harry N. Abrams Inc. \(1989\) ISBN:0810917092](#)
- EN ISO 13788: fRsi and Vapour Diffusion
- EN ISO 13789: references to Psi and Chi coefficients (inner/outer lengths)
- EN ISO 14683: Thermal Coupling Coefficient L^{nd} sometimes shown as H_D - direct heat transmission coefficient, references to Psi and Chi coefficients
- EN ISO 14683 Annex A: Thermal Bridges reference

The book "Thermal Heat Bridges" by W.Heindl, K.Krec, E.Panzhauser, A.Sigmund

This publication is available in German only.

Das Buch "Wärmebrücken" von W.Heindl, K.Krec, E.Panzhauser, A.Sigmund

Das Buch präsentiert neue einfache Methoden für den Hochbauingenieur zur Berücksichtigung der Auswirkungen von Wärmebrücken. Im Vordergrund steht dabei die für die Planung von Hochbauten wichtige Frage, ob, wo und unter welchen Bedingungen Oberflächenkondensation zu erwarten ist. Auch die nicht minder wichtige Frage nach den durch Wärmebrücken verursachten Wärmeverlusten wird beantwortet. Dabei wird besonderen Wert darauf gelegt, dass die durch die EDV-Berechnungen ermittelten Wärmebrücken-Kennwerte anschaulich fassbar sind und als Grundlage einfacher Auswertungen ohne weiteren Einsatz der EDV dienen können; bei diesen Auswertungen können Wärmeverluste, Oberflächentemperaturen und kritische relative Luftfeuchtigkeiten für beliebige Außen- und Innenlufttemperaturen mittels einfacher Formeln berechnet bzw. aus einer Tabelle abgelesen werden.



Das Buch bietet eine praxisorientierte Einführung in die für die Beurteilung von Wärmebrücken wichtigsten Begriffe, Formeln und Rechenmethoden. Im Hochbau häufig auftretende Wärmebrückeneffekte werden anhand von Beispielen erläutert. Ein umfangreicher Baudetailkatalog wird dem Planer das Gefühl für wärmetechnisch zweckmäßige Lösungen vermitteln, so dass er besser entscheiden kann, in welchen Fällen eine genaue Berechnung der Kennwerte seiner Konstruktion mittels EDV unentbehrlich ist.

Die theoretischen Grundlagen der angewandten Methoden und ein Exkurs über die experimentelle Entwicklung der Kennwerte von Wärmebrücken runden das Werk ab. Damit wird das Buch Architekten und Bauphysikern nicht nur als direkte Hilfe bei der Planung von Nutzen sein, sondern auch den an der Theorie Interessierten Einblick in die grundlegenden Zusammenhänge gewähren.

This book can be ordered (for example) at [amazon.de](https://www.amazon.de) : [W. Heindl, K. Kreč, E. Panzhauser & A. Sigmund, "Wärmebrücken", Springer-Verlag \(1987\) ISBN:3211820248, ISBN:0387820248](https://www.amazon.de)

See also: [Theoretical background](#)

On condensing humidity (dew point)

A practical measurement indicator for the amount of vapour contained within the wet air of some specific temperature is given by the relative humidity. It can be measured directly and very precisely with simple instruments - e.g. the hair hygrometer. When wet air wipes some surface of which the temperature is lower than that of the air, then it will show the vapour condensation on that surface when its temperature falls below the dew point temperature of the air. The dew point temperature is set by the value of temperature at which wet air of some specific vapour content will be saturated (by holding other parameters constant), i.e. the relative humidity is 100%.

The dew point temperature is simultaneously the degree of air moisture. Unfortunately it is not easy to measure (e.g. with a dew point thermometer) and by that it has not been adopted in practical use. De facto the dew point provides an ideal indicator to assess condensation risk at building component surfaces. If the dew point temperature is below the actual surface temperature there will be no condensation. To reliably preclude condensation the temperature at any place of the interior surface must be higher than the current dew point temperature of the air.

In practice, however, one will rarely have an idea about the dew point, rather than, on the other hand, about the temperature and relative air humidity. For that reason the term of "condensing humidity" is introduced when assessing surface temperatures in the context of condensation risk.

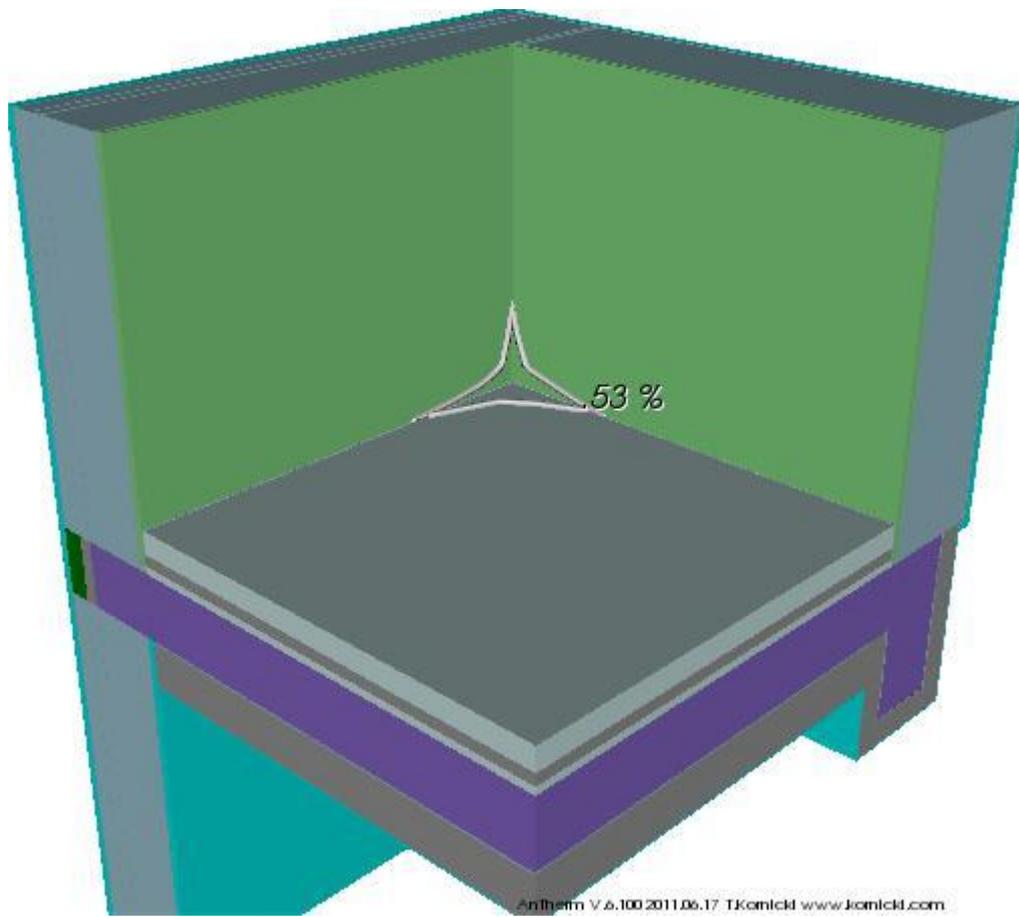
The "condensing humidity" is the value of relative air humidity at which, when it is exceeded, the vapour condensation will occur on the surface having some specific temperature. It can be calculated by a quotient of saturated vapour pressure at surface temperature and the pressure of saturation at specific air temperature.

The introduction of condensing humidity provides direct normative assessment of the temperature field calculated for some construction. The ÖNorm B8110-2:2003 defines the standard interior climate for residential spaces as follows: 20 °C air temperature and 65% relative air humidity when exterior temperature is 0° C. For negative temperatures of exterior air the maximum allowed humidity for the interior space air has to be reduced by 1% by each degree of exterior air temperature decrease. The condensation risk must be concluded only when the "condensing humidity" calculated for the temperature of the coldest point at the surface goes below the one set within the standard.

Example: When the standardized temperature of exterior air is set for the calculation to -12°C, then the relative humidity of the interior air (at 20°C temperature) shall never go above $65 - 12 = 53\%$.

If the "condensing humidity" calculated for the coldest surface point is above 53%, then, concluding the ÖNorm B8110-2:2003, there is no condensation risk.

If this is not the case, the construction assessed must be considered non conformant with standards. The areas affected by the condensation risk can be easily made visible by drawing the [isolinie](#) of 53% of [condensing humidity](#) at that surface.



See also: [Condensing Humidity Calculator \(Tool\), Results report, Selected evaluation function](#)

Notes on calculating equivalent thermal transfer coefficient of air cavities

This essay is available in German only.

Notiz zur Berechnung äquivalenter Wärmeleitfähigkeiten für Hohlräume

Modul zur Berechnung äquivalenter Wärmeleitfähigkeiten für Hohlräume

Im Menü „Werkzeuge“ wird ein Programm zur Berechnung von äquivalenten Wärmeleitfähigkeiten für geschlossene Hohlräume angeboten werden.

Ein Hohlraum wird nur in Ausnahmefällen aus einem einzigen Element bestehen; im Normalfall wird ein Hohlraum durch eine Gruppe von Elementen beschrieben werden.

Die im Folgenden angeführten Berechnungsvorschriften orientieren sich an den Euronormen EN ISO 6946 und EN ISO 10077-2 und sind für zweidimensionale Berechnungen ausgelegt.

Ao. Univ. Prof. Dipl.-Ing. Dr. techn. Klaus Kreč, Büro für Bauphysik, Schönberg am Kamp, Österreich

Lesen Sie hier weiter:

- [Notiz zur Berechnung äquivalenter Wärmeleitfähigkeiten für Hohlräume \(pdf\)](#) 

The above essay has been incorporated with friendly permission from *Klaus Kreč, Büro für Bauphysik, A-3562 Schönberg am Kamp, Veltlinerstr. 9, Österreich/Austria* .

See also: [Theoretical background](#)

On calculation of Ψ -values (Psi) for a 3 space case: Special consideration: Unheated spaces

This essay is available in German only.

Notiz zur Berechnung von Ψ -Werten (Psi) im 3-Raum-Fall; Spezialfall: unbeheizte Pufferräume

Eine Notiz zur ψ -Wert Berechnung im 3-Raum-Fall.

Die Problematik wird im Zusammenhang mit den bodenberührten Bauteilen behandelt (unbeheizter Kellerraum) und das Thema ist nun ganz allgemein abgearbeitet. Aus mehreren Anfragen zeigt sich, dass die Problematik schnell zur Konfusion und zur (unsinnigen) physikalischen Interpretation von ψ -Werten (ver)führt.

Continue reading here:

- [\$\psi\$ -Werte im 3-Raum-Fall; Spezialfall: unbeheizte Pufferräume](#) 

The above essay has been incorporated with friendly permission from *Klaus Kreč, Büro für Bauphysik, A-3562 Schönberg am Kamp, Veltlinerstr. 9, Österreich/Austria*

See also: [Linear and Point Transmittance, Theoretical background](#)

On calculation of Ψ -values (Psi) for building constructions in connection with ground

This essay is available in German only.

Zur Berechnung von Ψ -Werten (Psi) für Baukonstruktionen im Bereich bodenberührter

Bauteile

Die „indirekte Methode“ zur Berechnung des thermischen Leitwerts der Gebäudehülle nach EN ISO 10211 basiert auf einem eindimensionalen Berechnungsansatz, der im Nachhinein durch Aufsummierung sog. „Leitwertzuschläge“ korrigiert wird. Die Leitwertzuschläge werden jeweils durch Bildung des Produkts aus dem „ Ψ -Wert“ (Psi Wert) und der zugehörigen Länge errechnet.

Da die „indirekte Methode“ mehrdeutig und nicht für jeden Spezialfall normativ geregelt ist, führt sie immer wieder zu Verunsicherung und in der Folge auch zu – durchaus auch schwerwiegenden – Fehlern. Im Fall bodenberührter Bauteile, für den ja eine eindimensionale Modellierung von vornherein fragwürdig erscheint, ist die Verunsicherung besonders ausgeprägt. Daher soll hier ein Versuch der Klärung anstehender Fragen unternommen werden. Hierbei werden allgemein gültige, wesentliche Grundsätze untersucht und dargelegt. Die oft gewünschte kochbuchartige Beschreibung einer einzuhaltenden Vorgangsweise könnte lediglich Spezialfälle abdecken und wird hier bewusst vermieden.

Von grundsätzlicher Bedeutung ist die Einsicht, dass die „indirekte Methode“ eine reine Rechenvorschrift zur Korrektur des Fehlers der eindimensionalen Modellierung ist. Daher ist es in der überwiegenden Zahl der Fälle auch unzulässig und irreführend, den mittels zweidimensionalen Wärmebrückenberechnungen ermittelten „ Ψ -Werten“ irgendeine physikalische Bedeutung beizumessen. Zu betonen ist allerdings, dass die „indirekte Methode“ bei richtiger Anwendung zum gleichen Resultat – dem thermischen Leitwert der Gebäudehülle – führt wie die „direkte Methode“ der EN ISO 10211 oder eine dreidimensionale Modellierung des gesamten Gebäudes.

Die Mehrdeutigkeit der „indirekten Methode“ wird durch den Umstand hervorgerufen, dass das verwendete eindimensionale Berechnungsmodell zur Beantwortung der Problemstellung (Berechnung des Leitwerts der Gebäudehülle, Heizwärme- und Heizenergiebedarfsberechnung, Sommertauglichkeitsberechnung, ...) grundsätzlich nicht geeignet ist und auf verschiedenste Weise konstruiert werden kann. Der Fehler der eindimensionalen Berechnung ist somit eng mit der Art des gewählten eindimensionalen Modells verbunden. Die Korrektur dieses Fehlers mittels Leitwert-Zu- oder Abschlägen wird dann und nur dann richtige Resultate liefern, wenn die Berechnung der Ψ -Werte und der Längen unter Zugrundelegung des exakt gleichen eindimensionalen Modells erfolgt wie die Berechnung des Gesamtleitwerts.

Die nachfolgende Untersuchung des Spezialfalls bodenberührter Bauteile fußt auf den normativen Vorgaben der EN ISO 10211, der EN ISO 13370 und der ÖNorm B8110-6.

Continue reading here:

- [Zur Berechnung von \$\Psi\$ -Werten für Baukonstruktionen im Bereich bodenberührter Bauteile \(pdf\)](#) 

The above essay has been incorporated with friendly permission from Klaus Kreč, Büro für Bauphysik, A-3562 Schönberg am Kamp, Veltlinerstr. 9, Österreich/Austria

See also: [Linear and Point Transmittance, Theoretical background](#)

Temperature factor fRsi (f_{Rsi}^*)

Thermal bridges cause additional heat losses on one hand and on the other they cause low interior surface temperatures. Accordingly, to characterize the effect of thermal bridges, two different, independent parameters are needed - f_{Rsi}^* and thermal coupling coefficient (Leitwert, thermal conductance). For the case of many spaces connected via the thermal bridge these are the weighting factors g_0, \dots, g_n and the thermal conductance matrix of thermal coupling coefficients L_{ij} .

To estimate the risk of a possible condensation or mould growth, the surface temperature of the interior space must be known. It is not reasonable to declare this temperature in degrees of Celsius, as this would only be valid for some defined boundary conditions mix, i.e. some specific temperature set for indoor air space and exterior air. Instead the temperature factor f_{Rsi}^* , as defined in [EN ISO 10211](#), shall be used (in the [bibliography](#) is also known as temperature difference quotient Θ).

It is a constant, construction specific value, that is independent of any temperature difference between indoor and outdoor climate. This factor can be calculated only with two- and three dimensional heat transport simulation software.

Thanks to the [super-refinement](#) AnTherm evaluates temperature factors f_{Rsi}^* **directly out of base solutions (the weighting factors) and independently of the dimensionality of thermal bridge (2D or 3D).**

The temperature factor f_{Rsi}^* is calculated and output in the [results report](#) for the coldest point of any respective space - in addition to the weighting factors. For the case of only two space it is nothing else but $1 - g_{Se}$ (i.e. within results report the row of weighting factors for the exterior space shall be subtracted from 1).

Important: The temperature factor f_{Rsi}^* defines [for only two spaces connected to the thermal bridge](#). When three or more spaces are considered the temperature factors g_0, \dots, g_n must be used for the evaluation!

Definition

The temperature factor f_{Rsi}^* (fRsi): the difference between the interior surface temperature θ_{si} of a component and exterior air temperature θ_e , related to (divided by) the difference of temperatures between interior air θ_i and exterior air θ_e . The surface temperature is to be determined with some well defined surface resistance R_{si} :

$$\Theta = f_{Rsi}^* = (\theta_{si} - \theta_e) / (\theta_i - \theta_e)$$

where

θ_{si} [°C] interior surface temperature

θ_e [°C] exterior air temperature

θ_i [°C] interior air temperature

The calculation of the interior surface temperature °C can be then derived from following equation:

$$\theta_{si} = f_{Rsi}^* \cdot (\theta_i - \theta_e) + \theta_e$$

When one dimensional case is considered, e.g. to asses regular layer structure of external wall, the f_{Rsi}^* might be calculated from

$$f_{Rsi}^* = 1 - U / \alpha_{si}$$

or

$$f_{Rsi}^* = 1 - U \cdot R_{si}$$

where

U [W/(m·K)] thermal conductance of the wall

α_{si} [W/(m²·K)] thermal surface conductance of the interior surface

R_{si} [m²·K/W] thermal resistance of the interior surface.

Taking the definition formula of f_{Rsi}^* and replacing the terms with the formula used within AnTherm $\theta_{si} = \theta_i \cdot g_{Si} + \theta_e \cdot g_{Se}$ (valid for the two space case only!), one shall obtain as intermediate result:

$$f_{Rsi}^* = ((\theta_i \cdot g_{Si} + \theta_e \cdot g_{Se}) - \theta_e) / (\theta_i - \theta_e)$$

Considering simultaneously, that for the two space case also $g_{Si} + g_{Se} = 1$ is valid, thus $g_{Se} = 1 - g_{Si}$ holds too, one receives short equation:

$$f_{Rsi}^* = 1 - g_{Si}$$

or

$$f_{Rsi}^* = g_{Se}$$

Thanks to the [super-refinement](#) AnTherm evaluates temperature factors f_{Rsi}^* directly and precisely out of base solutions (the weighting factors) and independently of the dimensionality of thermal bridge (2D or 3D).

Space corners in three dimensions

At corners of spaces there are three two dimensional thermal bridges crossing over which consequently results in three dimensional distribution of temperatures. At such corners lower interior temperatures result when compared to those of two dimensional thermal bridges (edges). Therefore significantly higher risk of mould growth or vapour condensation is located here.

Thanks to the [super-refinement](#) AnTherm evaluates temperature factors f_{Rsi}^{3D} directly and precisely out of base solutions (the weighting factors) and independently of the dimensionality of thermal bridge (2D or 3D).

Minimum requirements ([design factors](#))

To reduce the mould growth risk by applying design measures several requirements shall be fulfilled.

For example, for all structural, geometrical and material thermal bridges deviating from DIN 4108 Beiblatt 2, the temperature factor f_{Rsi}^* at the worst location must suffice the requirement of $f_{Rsi}^* \geq 0,70$.

The respective [design factors](#) are specified in local regulations of [the assessment of condensation or mould growth risk](#) and vary strongly from country to country.

See also: [Results report, Bibliography](#), EN ISO 13788

Notes on calculation methods used in AnTherm to describe multidimensional vapour diffusion process



This essay is available in German only.

Update: The equation (14) of the EN ISO 13788 defines a factor ("water vapour diffusion conductance coefficient") equal to $2 \cdot 10^{-10}$ kg/Pa*m*s. If converted to the units used in AnTherm it leads to a factor 1,3888 (instead of 1,5 following earlier standards). To stay conformant with current standards the factor 1,5 was changed to 1,3888 - or better to 10/7,2. One shall expect only marginal changes to the results!

Notiz zu den von AnTherm verwendeten Berechnungsgrundlagen zur Beschreibung mehrdimensional ablaufender Wasserdampf-Diffusionsvorgänge

Da das hier skizzierte physikalische Modell streng analog zum in AnTherm umgesetzten Modell zur Beschreibung mehrdimensionaler Wärmeströme ist, kann AnTherm unmittelbar auch zur Beschreibung von mehrdimensional ablaufenden Diffusionsvorgängen von Wasserdampf eingesetzt werden. Es ist lediglich die ...

Ao. Univ. Prof. Dipl.-Ing. Dr. techn. Klaus Kreč, Büro für Bauphysik, Schönberg am Kamp, Österreich

Update: In der EN ISO 13788 ist in Gleichung (14) ein Faktor ("Wasserdampfdiffusionsleitkoeffizient") definiert ($2 \cdot 10^{-10}$ kg/Pa*m*s), der umgerechnet auf das im AnTherm verwendete Einheitensystem auf den Faktor 1,3888 (anstelle von 1,5 nach früheren Normen) führt. Um konform mit den aktuellen Normen zu sein Faktor 1,5 auf 1,3888 - bzw. besser auf 10/7,2 - gesetzt. Durch diese Änderung sind keine großen Veränderungen im Ergebnis zu erwarten!

Lesen Sie hier weiter:

- [Notiz zu den von AnTherm verwendeten Berechnungsgrundlagen zur Beschreibung mehrdimensional ablaufender Wasserdampf-Diffusionsvorgänge \(pdf\)](#) 



The above essay have been incorporated with friendly permission from *Klaus Kreč, Büro für Bauphysik, A-3562 Schönberg am Kamp, Veltlinerstr. 9, Österreich/Austria* .

See also: [Theoretical background](#)

The air temperature in the unheated basement, a case study

This essay is available in German only.

Die Lufttemperatur im unbeheizten Kellergeschoß; eine Fallstudie

Der Jahresverlauf der sich in einem unbeheizten Kellergeschoß einstellenden Lufttemperatur wird für verschiedenste Dämm-Varianten unter Zugrundelegung eines dreidimensionalen, zeitlich periodisch eingeschwungenen thermischen Modelles berechnet. Die Untersuchung der Wirksamkeit von Dämm-Maßnahmen im Bereich der Kellerdecke führt zum Ergebnis, dass die Reduktion der Wärmeverluste an den unbeheizten Kellerraum aufgrund von Zusatzdämmung wesentlich kleiner ausfällt als es bei Verwendung der üblicherweise angewandten eindimensionalen rechnerischen Abschätzungen zu erwarten wäre.

Die Kenntnis des Jahresverlaufes der Raumlufttemperatur eines nicht beheizten Kellergeschoßes ist im allgemeinen aus mehreren Gründen von Bedeutung. Zum einen bestimmt die Temperaturdifferenz zwischen dem über dem Keller gelegenen, beheizten Geschoß und dem Kellergeschoß den Transmissionswärmeverlust durch die Kellerdecke und hat damit auch Einfluss auf den Heizenergiebedarf für das beheizte Gebäudevolumen. Zum anderen ist die Temperatur im Kellergeschoß entscheidend für die Möglichkeiten, dieses zu nutzen. So ist es interessant zu wissen, inwieweit der Keller zur Lagerung verderblicher Güter - z. B. von Lebensmitteln - geeignet ist. Insbesondere kommt auch der Frostsicherheit im Winter große Bedeutung zu.

Zudem kann es gerade in unbeheizten Kellerräumen zu Oberflächenkondensation und in der Folge zu Schimmelbildung kommen, wenn die Oberflächentemperaturen der bodenberührten Bauteile über längere Zeit stark unter die Raumlufttemperatur absinken. Aus den genannten Gründen ist es erstrebenswert, den im langjährigen Schnitt zu erwartenden Jahresverlauf der Lufttemperatur in einem unbeheizten Kellergeschoß und die Temperaturen der dem Kellergeschoß zugewandten Bauteiloberflächen berechnen zu können.

Die hier vorliegende Arbeit befasst sich ausschließlich mit der erstgenannten Problemstellung. Die Schilderung der auf die Ergebnisse der vorliegenden Untersuchung aufbauenden Berechnung der Oberflächentemperaturen soll einer späteren Publikation vorbehalten bleiben.

...Die Berechnung des Jahresverlaufes der Lufttemperatur innerhalb eines unbeheizten Kellergeschoßes läuft auf die instationäre, zeitlich periodisch eingeschwungene Simulation des thermischen Verhaltens eines Raumes unter Berücksichtigung dreidimensional ablaufender Wärmeleitungs- und Wärmespeicherungsvorgänge hinaus. Das hier vorgestellte Simulationsmodell ist insofern vereinfacht, als etwaige Innenwärmen im Kellergeschoß vernachlässigt werden. Die Berücksichtigung von Innenwärmen bereitet - wie in [7] gezeigt - keine grundsätzlichen Schwierigkeiten, sodass auch die Entwicklung eines dreidimensional, periodisch eingeschwungen arbeitenden Rechenprogrammes zur Beschreibung des thermischen Verhaltens ganzer Gebäude prinzipiell realisierbar erscheint.

Continue reading here:

- [Die Lufttemperatur im unbeheizten Kellergeschoß; eine Fallstudie \(pdf\)](#)



The above essay has been incorporated with friendly permission from *Klaus Kreč, Büro für Bauphysik, A-3562 Schönberg am Kamp, Veltlinerstr. 9, Österreich/Austria* .

[7] K. Kreč, "Wärmeleitung in Baukonstruktionen unter Berücksichtigung von Wärmequellen", Gesundheitsingenieur 114, Heft 6, 313-318, 1993

See also: [Theoretical background](#)

Heat losses from large buildings through the ground

The heating demand of a building is strongly influenced by transmission heat losses through the ground, if the total thermal conductance of the building constructions in contact with the ground is in the same order as the thermal conductance of the building envelope in contact with the air. This will be the case in particular for such buildings, where the building constructions in contact with the ground take up a considerable part of the total area of the building envelope. On the one hand this situation is given for small buildings provided with a basement. On the other hand transmission heat losses through the ground are important for buildings with big floor areas in direct contact to the ground, e.g. for big halls.

Transmission heat losses from a basement of a residential building were investigated in [1] and [2]. This paper deals with heat losses through various slab on ground constructions normally used for big industrial halls.

Calculations of the heat flow through building constructions in contact with the ground are special cases on principle in that they cannot be done using the ordinary constant, one-dimensional (1D) thermal model. On the one side the heat flow through the ground can be described only using two or three spatial coordinates. From this heat losses through the ground are to be calculated using two- and three-dimensional (2D and 3D) thermal models. On the other side in most cases of interest the very big heat storage capacity of the ground involved in such heat flow calculations cannot be ignored, so that a non-steady calculation method is required.

For most slab shaped building constructions in contact with the air the thermal quality of the construction with regard to heat losses can be characterized by an U-value in good approximation. Contrary to this the declaration of a U-value alone makes no sense for slab on ground constructions. The well known fact that the heat loss through a slab-shaped building construction can be calculated in good approximation by multiplying the U-value of the slab with it's area and with the difference of the air temperatures between internal and external environment is no longer true for slab on ground constructions. The heat flow through slabs on ground is not only influences by the U-value and the area of the slab but also by the geometry of the building construction in contact with the ground and by the thermal characteristics of the surrounding soil. For that reason quantitative statements concerning the heat through the ground are only true for the specific building under consideration.

In this paper a case study concerning different insulation levels of a slab on ground construction is performed for an industrial hall with largely fixed geometry. From this study qualitative conclusions concerning the effectiveness of slab insulation is drawn.

... The limitation of the heating demand of industrial halls often requires a reduction of the transmission heat losses through the ground. Generally calculations of heat losses through the ground require a three-dimensional, non-steady treatment. ...

Continue reading here:

- [Heat losses from large buildings through the ground \(pdf\)](#)



The above essay have been incorporated with friendly permission from *Klaus Kreč, Büro für Bauphysik, A-3562 Schönberg am Kamp, Veltlinerstr. 9, Österreich/Austria* .

[1] K. Kreč, "Wärmeverluste über erdbodenberührte Bauteile Fallstudie beheizter Kellerraum", WKS 33,32-35 (1993)

[2] [K. Kreč, "Lufttemperatur im unbeheizten Kellergeschoß; eine Fallstudie", WKS 34,42-47 \(1994\)](#)

See also: [Theoretical background](#)

Methods of thermal heat bridge survey in international perspective

This essay is available in German only. Translation of the abstract:

The recording of the effect of thermal bridges on the thermal behaviour of building constructions, rooms, room groups and buildings is becoming increasingly important in the course of the continuously increasing demand for thermal quality. The reason for this is the fact that the one-dimensional calculation of temperature distributions and heat flows in building constructions with well insulated construction methods, as they are state of the art today, delivers too inaccurate results. The effects of such oversimplified calculation methods can be misjudgments regarding the heating design and the expected heating energy demand, but also the overlooking of existing condensation risks. A metrological recording of the behaviour of building constructions with thermal bridges fails in particular with larger components or whole building parts because of the necessary effort.

The importance of a sufficiently precise recording of thermal bridges, i.e. areas in which thermal conduction and storage effects can only be calculated using a 2- or 3-dimensional thermal model, has been taken into account since the beginning of the 1980s insofar as a large number of proposals have been discussed with regard to the introduction of correction factors for the recording of thermal bridges, programs for the simulation of 2- and 3-dimensional thermal processes have been created and offered and, finally, several working groups (TC 89: WG1, WG4 and WG5) have also addressed this problem in the course of European standardisation.

In the following, a physically founded theoretical model for recording the thermal behaviour of building structures with thermal bridges will be presented and the implementation of this theoretical basis in practical use and the relevant provisions of European standardization (TC 89) will be discussed.

Methoden der Erfassung von Wärmebrücken im internationalen Vergleich

Der Erfassung der Auswirkung von Wärmebrücken auf das thermische Verhalten von Baukonstruktionen, Räumen, Raumgruppen und Gebäuden kommt im Zuge der laufend größer werdenden Anforderung an die thermische Qualität stark steigende Bedeutung zu. Grund hierfür ist der Umstand, dass die bisher eingebürgerte eindimensionale Berechnung von Temperaturverteilungen und Wärmeströmen in Baukonstruktionen bei gut gedämmten Bauweisen, wie sie heute Stand der Technik sind, zu ungenaue Ergebnisse liefert. Die Auswirkungen solch' übersimplifizierter Berechnungsmethoden können Fehleinschätzungen bezüglich der Heizungsauslegung und des zu erwartenden Heizenergiebedarfes aber auch das Übersehen bestehender Kondensationsrisiken sein. Eine messtechnische Erfassung des Verhaltens von Baukonstruktionen mit Wärmebrücken scheidet insbesondere bei größeren Bauteilen oder ganzen Gebäudeteilen am hierfür notwendigen Aufwand.

Der Bedeutung einer genügend genauen Erfassung von Wärmebrücken, also Bereichen in denen Wärmeleitungs- und Wärmespeicherungseffekte nur unter Verwendung eines 2- oder 3-dimensionalen thermischen Modells berechnet werden können, wurde insofern seit Anfang der 80er Jahre Rechnung getragen, als eine Vielzahl von Vorschlägen bezüglich der Einführung von Korrekturfaktoren zur Erfassung von Wärmebrücken diskutiert wurde, Programme zur Simulation von 2- und 3-dimensional ablaufenden thermischen Vorgängen geschaffen und angeboten wurden und sich schließlich auch im Zuge der europäischen Normung etliche Arbeitsgruppen (TC 89: WG1, WG4 und WG5) dieser Problematik annahmen.

Im Folgenden wird ein physikalisch fundiertes theoretisches Modell zur Erfassung des thermischen Verhaltens von Baukonstruktionen mit Wärmebrücken vorgestellt und auf die Umsetzung dieser theoretischen Grundlage im Praxiseinsatz sowie auf die diesbezüglich im Rahmen der europäischen Normung (TC 89) gesetzten Festlegungen eingegangen.

Lesen Sie hier weiter:

- [Methoden der Erfassung von Wärmebrücken im internationalen Vergleich \(pdf\)](#)



The above essay have been incorporated with friendly permission from Klaus Kreč, Büro für Bauphysik, A-3562 Schönberg am Kamp, Veltlinerstr. 9, Österreich/Austria .

See also: [Theoretical background](#)

On three dimensional simulation of thermal behaviour of buildings

This essay is available in German only. Translation of the abstract:

The concept of harmonic thermal coupling coefficients is used to describe the multidimensional heat conduction processes taking place in the components of a building, taking heat storage into account. The design of a thermal network leads to an easily comprehensible concept for multidimensional programs for the simulation of the thermal behaviour of buildings. The possibilities and limits of such a simulation program are shown using the example of a residential building with a highly glazed south façade that is lowered into the ground.

The simulation of the thermal behaviour of buildings requires, among other things, the computational recording of heat conduction and heat storage processes in the building components. In the last three decades, various calculation algorithms have been developed for this purpose - see e.g. Gertis & Hauser [1] or Feist [2]. However, the multitude of methods and programs for recording the thermal behaviour of buildings has one thing in common. The heat conduction equation is applied and solved in one-dimensional approximation. At best, multidimensional heat conduction processes are taken into account by introducing stationary correction factors.

The necessity of considering areas of multi-dimensional heat conduction - usually referred to as "thermal bridges" - is not disputed for buildings that are thermally insulated according to today's building codes or thermal insulation regulations. Thus, in national (see e.g. ÖNorm B8110-1 [3]) as well as international standards [4], the consideration of thermal bridges in the calculation of the heating demand of buildings is mandatory. The calculation approaches specified in these standards are limited to stationary, i.e. time-independent, processes. Heat storage processes that influence the multidimensional heat conduction processes are not taken into account.

This unsatisfactory situation was countered with the development of a theoretical concept that allows transient multidimensional heat conduction processes to be recorded taking into account heat storage [5, 6]. In the meantime, the concept of thermal coupling coefficients developed by Heindl et al. [7] has been anchored in the European standard on the mathematical treatment of thermal bridges [8]. The generalization of this concept to transient processes leads according to [5] to complex thermal coupling coefficients. The generalized conductance concept was at least taken over from the general approach in the international standard EN ISO 13786 [9]. The "generalized conductance" introduced in [5] is referred to in EN ISO 13786 as the "harmonic thermal conductance" - a term that is also to be used here.

In the present work, a calculation model for a multi-dimensional simulation program for recording the thermal behaviour of buildings is presented and discussed with access to the theoretical principles developed in [7], [5] and [6].

... Programs which are based on the described physical model allow the multidimensional simulation of the thermal behaviour of buildings and are characterized by extremely short computation times and immediately interpretable results. The handling of the simulation module can be designed extremely simply for the user, since the three-dimensional modelling of the building, which is necessary for the computation of the conductance matrix and the heating power distribution key, takes place under access to a transient "thermal bridge program" in an advance computation. The results of the simulation - time curves of the indoor air temperatures and/or time curves of heating or cooling capacities - can be used to calculate the time curve of the temperature field within the building components by transferring them to the "thermal bridge program" used in the first step. Of course, it is also possible to call up the time course of the surface temperatures, which occurs taking into account the multidimensional heat conduction and heat storage processes.

The heat storage effects occurring in the components of a building influence the heat flow that occurs. In multidimensional thermal models, these heat storage effects are not taken into account in building simulation programs commonly used today. With the calculation model discussed here, it will be possible to quantify the magnitude of the error caused by such neglect. In particular for summer suitability calculations, the planning reliability can be increased by now possible examination of the usually used simplified approaches. In addition, it is now also possible - as shown in the example - to check and optimise those design strategies that are used for heat storage systems.

Zur dreidimensionalen Simulation des thermischen Verhaltens von Gebäuden

Das Konzept der harmonischen thermischen Leitwerte wird verwendet, um die in den Bauteilen eines Gebäudes mehrdimensional ablaufenden Wärmeleitungsvorgänge unter Berücksichtigung der Wärmespeicherung zu beschreiben. Der Entwurf eines thermischen Netzwerkes führt auf ein leicht überschaubares Konzept für mehrdimensional arbeitende Programme zur Simulation des thermischen Verhaltens von Gebäuden. Die Möglichkeiten und Grenzen eines derartigen Simulationsprogramms werden am Beispiel eines in den Erdboden abgesenkten Wohnhauses mit hochverglaster Südfassade aufgezeigt.

Die Simulation des thermischen Verhaltens von Gebäuden erfordert unter anderem die rechnerische Erfassung von Wärmeleitungs- und Wärmespeicherungsvorgängen in den Bauteilen des Gebäudes. In den letzten drei Jahrzehnten wurden hierfür verschiedenste Berechnungsalgorithmen entwickelt – siehe z. B. Gertis & Hauser [1] oder Feist [2]. Der Vielzahl an Methoden und Programmen zur Erfassung des thermischen Verhaltens von Gebäuden haftet jedoch eine Gemeinsamkeit an. Die Wärmeleitungsgleichung wird in eindimensionaler Näherung angesetzt und – meist näherungsweise – gelöst. Mehrdimensional ablaufende Wärmeleitungsvorgänge werden bestenfalls mittels Einführung von stationären Korrekturfaktoren berücksichtigt.

Die Notwendigkeit der Berücksichtigung von Bereichen mehrdimensionaler Wärmeleitung – schlagwortartig meist als "Wärmebrücken" bezeichnet – steht für Gebäude, die nach heutigen Bauordnungen oder Wärmeschutzverordnungen wärmedämmend sind, außer Streit. So ist in nationalen (siehe z. B. ÖNorm B8110-1 [3]) wie auch internationalen Normenwerken [4] die Berücksichtigung von Wärmebrücken bei der Berechnung des Heizwärmebedarfes von Gebäuden zwingend vorgeschrieben. Die in diesen Normen angegebenen Berechnungsansätze bleiben auf stationäre, d. h. zeitunabhängige Vorgänge beschränkt. Wärmespeicherungsvorgänge, die die mehrdimensional ablaufenden Wärmeleitungsvorgänge beeinflussen, bleiben unberücksichtigt.

Dieser unbefriedigenden Situation wurde mit der Entwicklung eines theoretischen Konzeptes, das instationär mehrdimensional ablaufende Wärmeleitungsvorgänge unter Berücksichtigung der Wärmespeicherung zu erfassen gestattet, begegnet [5, 6]. Mittlerweile ist das durch Heindl et al. [7] entwickelte Konzept der thermischen Leitwerte in der Europanorm über die rechnerische Behandlung von Wärmebrücken [8] verankert. Die Verallgemeinerung dieses Konzeptes auf instationäre Vorgänge führt gemäß [5] auf komplexwertige thermische Leitwerte. Das verallgemeinerte Leitwert-Konzept wurde zumindest vom allgemeinen Ansatz her in die internationale Norm EN ISO 13786 [9] übernommen. Der in [5] eingeführte "verallgemeinerte Leitwert" wird in der EN ISO 13786 als "harmonischer thermischer Leitwert" bezeichnet – ein Begriff, der auch hier verwendet werden soll.

In der vorliegenden Arbeit wird unter Zugriff auf die in [7], [5] und [6] entwickelten theoretischen Grundlagen ein Berechnungsmodell für ein mehrdimensional arbeitendes Simulationsprogramm zur Erfassung des thermischen Verhaltens von Gebäuden vorgestellt und diskutiert.

... Programme, die auf dem geschilderten physikalischen Modell aufbauen, erlauben die mehrdimensionale Simulation des thermischen Verhaltens von Gebäuden und zeichnen sich durch extrem kurze Rechenzeiten und unmittelbar interpretierbare Ergebnisse aus. Die Handhabung des Simulationsmoduls kann für den Benutzer äußerst einfach gestaltet werden, da die dreidimensionale Modellierung des Gebäudes, die zur Berechnung der Leitwert-Matrix und der Heizleistungsverteilungsschlüssel erforderlich ist, unter Zugriff auf ein instationäres "Wärmebrückenprogramm" in einer Vorwegberechnung erfolgt. Die Ergebnisse der Simulation - Zeitverläufe der Innenlufttemperaturen und/oder Zeitverläufe von Heiz- oder Kühlleistungen - können mittels Übergabe an das im ersten Schritt verwendete "Wärmebrückenprogramm" dazu verwendet werden, den zeitlichen Verlauf des Temperaturfeldes innerhalb der Bauteile des Gebäudes zu berechnen. Natürlich kann hierbei auch der Zeitverlauf der Oberflächentemperaturen, der sich unter Berücksichtigung der mehrdimensional ablaufenden Wärmeleitungs- und Wärmespeicherungsvorgänge einstellt, abgerufen werden.

Die in den Bauteilen eines Gebäudes ablaufenden Wärmespeicherungseffekte beeinflussen den sich einstellenden Wärmestrom. In mehrdimensionalen thermischen Modellen bleiben diese Wärmespeicherungseffekte bei heute gebräuchlichen Gebäudesimulationsprogrammen unberücksichtigt. Mit dem hier diskutierten Berechnungsmodell wird es möglich sein, die Größenordnung des Fehlers, der durch solch' eine Vernachlässigung entsteht, zu quantifizieren. Insbesondere für Sommertauglichkeitsberechnungen kann durch die nun mögliche Überprüfung der üblicherweise verwendeten vereinfachten Ansätze die Planungssicherheit erhöht werden. Zudem ist es nunmehr - wie im Beispiel gezeigt - möglich, auch jene Entwurfsstrategien zu überprüfen und zu optimieren, die Wärmespeicherfähigkeitseffekte auszunutzen versuchen.

Continue reading here:

- [Zur dreidimensionalen Simulation des thermischen Verhaltens von Gebäuden \(pdf\)](#)



The above essay has been incorporated with friendly permission from *Klaus Kreč, Büro für Bauphysik, A-3562 Schönberg am Kamp, Veltlinerstr. 9, Österreich/Austria* .

- [1] Gertis, K. und Hauser, G.: Instationäre Berechnungsverfahren für den sommerlichen Wärmeschutz im Hochbau. Verlag Ernst & Sohn, Berlin, Berichte aus der Bauforschung, Heft 103, (1976)
- [2] Feist, W.: Thermische Gebäudesimulation. C. F. Müller Verlag Heidelberg 1994
- [3] ÖNorm B8110-1, Wärmeschutz im Hochbau – Anforderungen an den Wärmeschutz und Nachweisverfahren. (2000)
- [4] EN 832, Wärmetechnisches Verhalten von Gebäuden – Berechnung des Heizenergiebedarfs – Wohngebäude (1998)
- [5] [Kreč, K.: On the Storage of Heat in Building Components \(Zur Wärmespeicherung in Baukonstruktionen\). Gesundheits-Ingenieur 114 \(1993\), H. 1, S. 11-18.](#)
- [6] [Kreč, K.: Wärmeleitung in Baukonstruktionen unter Berücksichtigung von Wärmequellen. Gesundheits-Ingenieur 114 \(1993\), H. 6, S. 313-318.](#)
- [7] [Heindl, W., Kreč K., Panzhauser E. und Sigmund A.: Wärmebrücken. Springer-Verlag Wien-New York 1987.](#)
- [8] EN ISO 10211-1: Wärmebrücken im Hochbau – Wärmeströme und Oberflächentemperaturen – Teil 1: Allgemeine Berechnungsverfahren. (1996)
- [9] EN ISO 13786: Wärmetechnisches Verhalten von Bauteilen – Dynamisch-thermische Kenngrößen – Berechnungsverfahren (1999)

See also: [Theoretical background](#)

On visualization of heat and vapour diffusion stream in three dimensional models

The difficulties in the depiction of the flow of heat and vapour through three-dimensionally modelled building constructions are taken up. A method well suited for practical use in finding regions which are critical with regard to heat and water vapour transport is presented. The pseudo colour image of the absolute values of the flow densities of heat and/or diffusion of water vapour immediately shows the regions of increased heat or vapour diffusion flow. Furthermore the plot of single flow lines clearly shows the three-dimensional path of the heat or water vapour through the building construction. The expressiveness of this method is demonstrated by an example representing a building construction which was analyzed by a three-dimensional calculation using the new thermal bridge program AnTherm.

This article is available in German only.

See also: [Theoretical background, http://dx.doi.org/10.1002/bapi.200690030](http://dx.doi.org/10.1002/bapi.200690030)

Zur Visualisierung von Wärme- und Wasserdampfdiffusionsstrom bei dreidimensionaler Modellierung

[Klaus Kreč & Tomasz Kornicki](#)

1. Einleitung

Bei der Berechnung des Wärmedurchgangs durch Baukonstruktionen hat sich im Fall zweidimensionaler Wärmeströmung seit langem die Darstellung des Wärmestroms mit Hilfe von Wärmestromlinien als wertvolles Hilfsmittel für den Bauplaner in der Baupraxis etabliert. Da ein Zusammenrücken der Wärmestromlinien Bereiche höherer Wärmestromdichte signalisiert, gibt ein Bild der Wärmestromlinien als eines der Ergebnisse einer Wärmebrückenberechnung auf einen Blick Aufschluss über die thermischen Schwachstellen der untersuchten Baukonstruktion. Das Bild der Wärmestromlinien ermöglicht es dem Bauplaner, rasch und effektiv sowohl Bauteilanschlüsse in Hinblick auf deren thermische Qualität zu optimieren als auch die richtigen Ansatzpunkte bei Sanierungsprojekten zu finden.

Bei dreidimensionaler Modellierung besteht das Problem, dass die Stromfunktion zwar nach wie vor berechenbar ist, die Visualisierung der nun räumlich verlaufenden Kurvenschar aber nicht zu einer einfach lesbaren Ergebnisdarstellung führt. Diese Problematik wurde bei der Neuentwicklung des Wärmebrückenprogramms AnTherm [1] aufgegriffen und unter Zugriff auf die heute verfügbaren Mittel der Darstellung dreidimensional modellierter Objekte einer praxistauglichen Lösung zugeführt.

2. Berechnung und Visualisierung der Wärmestromdichten

Wie die meisten Wärmebrückenprogramme verwendet auch AnTherm die Methode der finiten Differenzen zur Beschreibung dreidimensional ablaufender Wärmeleitungsprozesse. Der Bauteil wird somit vor der eigentlichen Berechnung automatisch in Quader zerlegt. Die Zahl der Quader steigt im Allgemeinen rasch mit der Größe des modellierten Bereichs und/oder der Feinheit der Modellierung an. Im Gegensatz zu zweidimensionalen Berechnungsfällen erreicht die Anzahl der zu bilanzierenden Zellen bei dreidimensionalen Berechnungsfällen schnell Werte von einer oder mehreren Millionen. Bei den heute bereits auf Standard-PC's verfügbaren Rechnerleistungen bereitet dieser Umstand jedoch keine Probleme mehr.

Nach der Berechnung steht mit den Temperaturen in den Quadermitten, sowie den Seiten- und Kantenmitten und den Ecken der Quader eine Beschreibung des dreidimensionalen Temperaturfelds in einem üblicherweise sehr feinen Netz zur Verfügung. Somit kann auch für jeden Punkt des Netzes der Vektor der Wärmestromdichte errechnet werden.

Um in einem ersten Schritt die Verteilung der Wärmestromdichte mit AnTherm anschaulich darzustellen, wird jedem Quader durch Mittelung ein Vektor der Wärmestromdichte zugeordnet und der Betrag dieses Vektors gebildet. Das Feld der Beträge der Wärmestromdichten kann nun in Form einer Falschfarbendarstellung anschaulich dargestellt werden. Ähnlich wie beim zweidimensionalen Fall besteht nun die Möglichkeit, Orte erhöhter Wärmestromdichte und somit thermische Schwachstellen der Baukonstruktion unmittelbar zu lokalisieren.

In einem zweiten Schritt kann der Benutzer einen Punkt an der Oberfläche oder innerhalb der Baukonstruktion definieren und das Auszeichnen der Wärmestromlinie durch diesen Punkt verlangen. AnTherm ermöglicht es, die Oberflächen transparent oder durchscheinend zu gestalten. Es ist somit möglich, in die dreidimensional modellierte Baukonstruktion „hinein zu schauen“ und den Weg der Wärme durch den Bauteil in leicht verständlicher und prägnanter Darstellung zu verfolgen.

3. Ein Beispiel

Die beschriebenen Möglichkeiten der Visualisierung dreidimensional verlaufender Wärmeströme soll im Folgenden an einem einfachen Beispiel demonstriert werden.

Der modellierte Fall greift die Problematik des Wärme- und Dampftransports in Dach und Wänden eines Hallenbads auf, das einerseits an eine Wohnung und andererseits an die Außenluft grenzt. Die modellierte Baukonstruktion wurde von einem in [\[2\]](#) beschriebenen zweidimensionalen Fall abgeleitet.

Die folgende, mit AnTherm generierte Skizze zeigt den untersuchten Ausschnitt aus der Gebäudehülle.

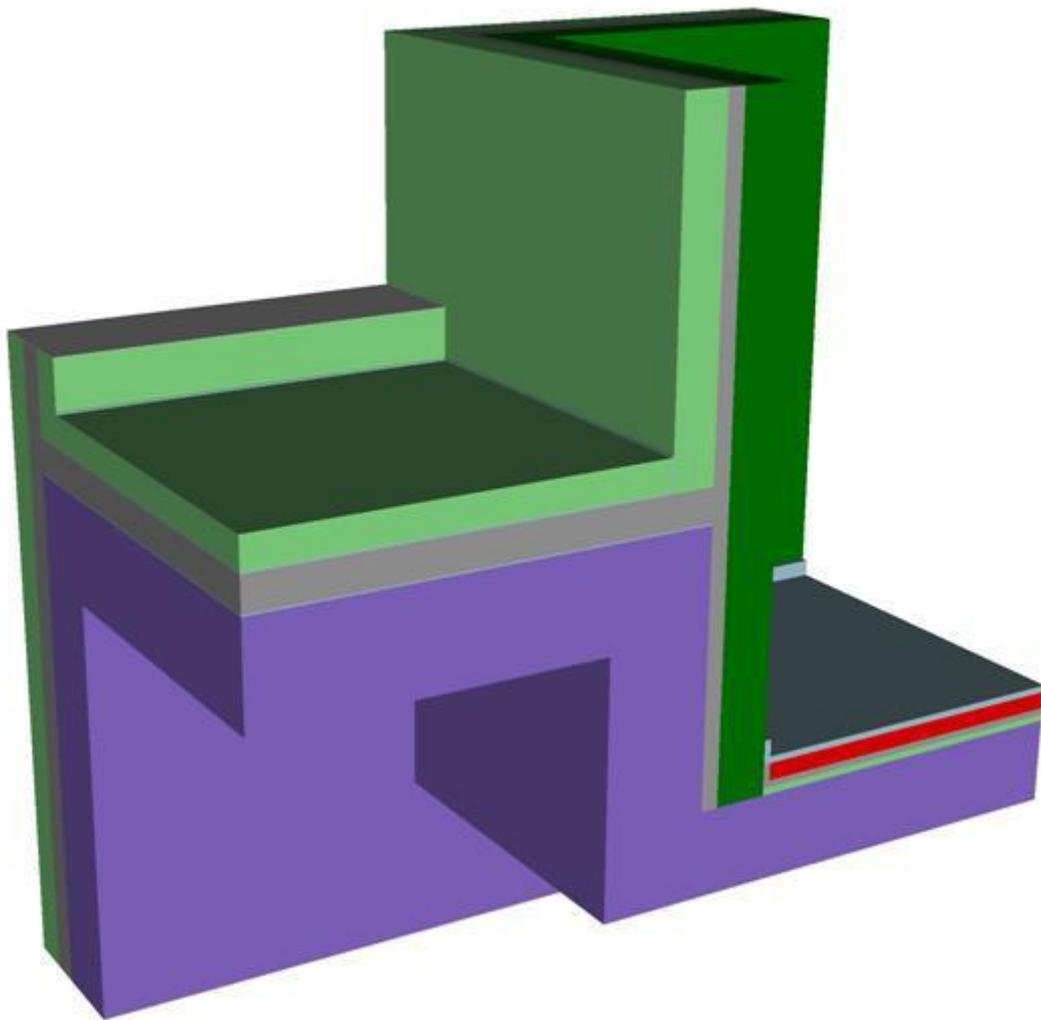


Abb. 1: Dreidimensionale Darstellung des modellierten Ausschnitts

Für das im unteren Bereich befindliche Hallenbad wurde bei den durchgeführten Berechnungen eine Lufttemperatur von 30 °C und eine relative Luftfeuchtigkeit von 80% angenommen. Auf der rechten Seite grenzt ein Wohnraum mit 20 °C und 53% r. F. direkt an das Hallenbad. Die thermische Verbindung nach außen ist im Hallenbad durch die rückseitig gelegene Außenwand (Betonwand mit außen liegender, 10 cm dicker Dämmschicht und Vorsatzschale aus Backstein) und das Flachdach gegeben. Der Wohnraum grenzt mit der rückseitig gelegenen Außenwand aber auch der über das Flachdach hinaus ragenden Außenwand nach außen. Diese Außenwand ist als zweischaliges Mauerwerk aus Backstein mit 5 cm Steinwolle als Kerndämmung ausgebildet. In der Berechnung wurde für die Außenluft eine Temperatur -12 °C und für die relative Luftfeuchtigkeit 80% angesetzt.

Die folgende Abbildung zeigt als Ergebnisdarstellung die Verteilung der Wärmestromdichte im modellierten Ausschnitt in Form eines Falschfarbenbilds.

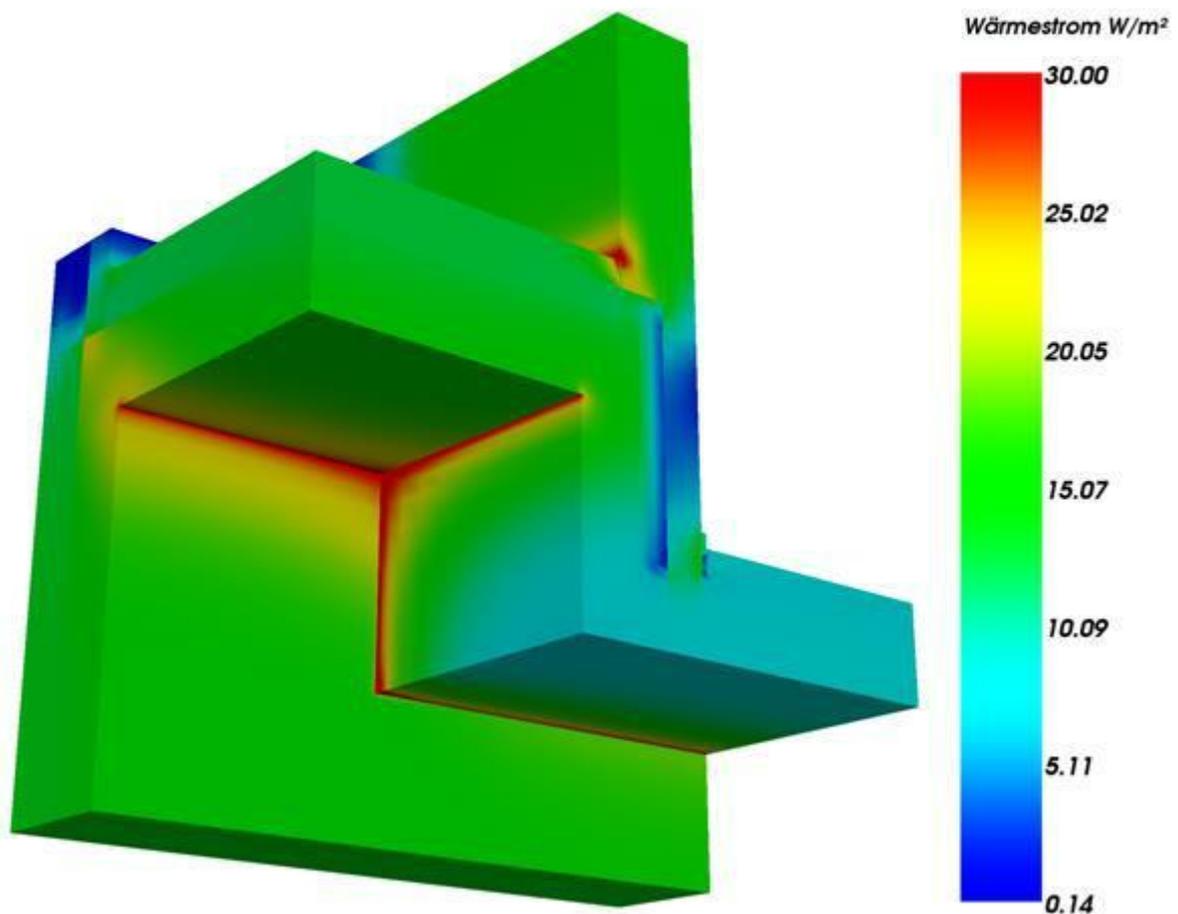


Abb. 2: Falschfarbenbild der Wärmestromdichte

Wie nicht anders zu erwarten, treten die größten Werte der im Falschfarbenbild ausgewiesenen Beträge der Wärmestromdichten in den Kanten und Ecken der raumbegrenzenden Bauteile des Hallenbads auf. Da die Grafik in AnTherm beliebig im Raum gedreht und somit von allen Seiten betrachtet werden kann, sind die thermischen Schwachstellen der untersuchten Baukonstruktion rasch lokalisiert.

Um geeignete Maßnahmen für eine Verbesserung der thermischen Qualität zu finden, ist es nun sehr wertvoll, den Weg der Wärme von einem als kritisch erkannten Punkt der inneren Oberfläche durch die Baukonstruktion zu kennen. Im Beispiel des Hallenbads soll der Weg der Wärme von der einspringenden Ecke nach außen visualisiert werden. Hierzu ist es zum einen notwendig, den Punkt, von dem aus eine Wärmestromlinie gezeichnet werden soll, festzulegen. Zum anderen muss die Oberfläche transparent gemacht werden, um in die Baukonstruktion hinein schauen zu können. Das mit Antherm generierte Ergebnis zeigt folgende Abbildung.

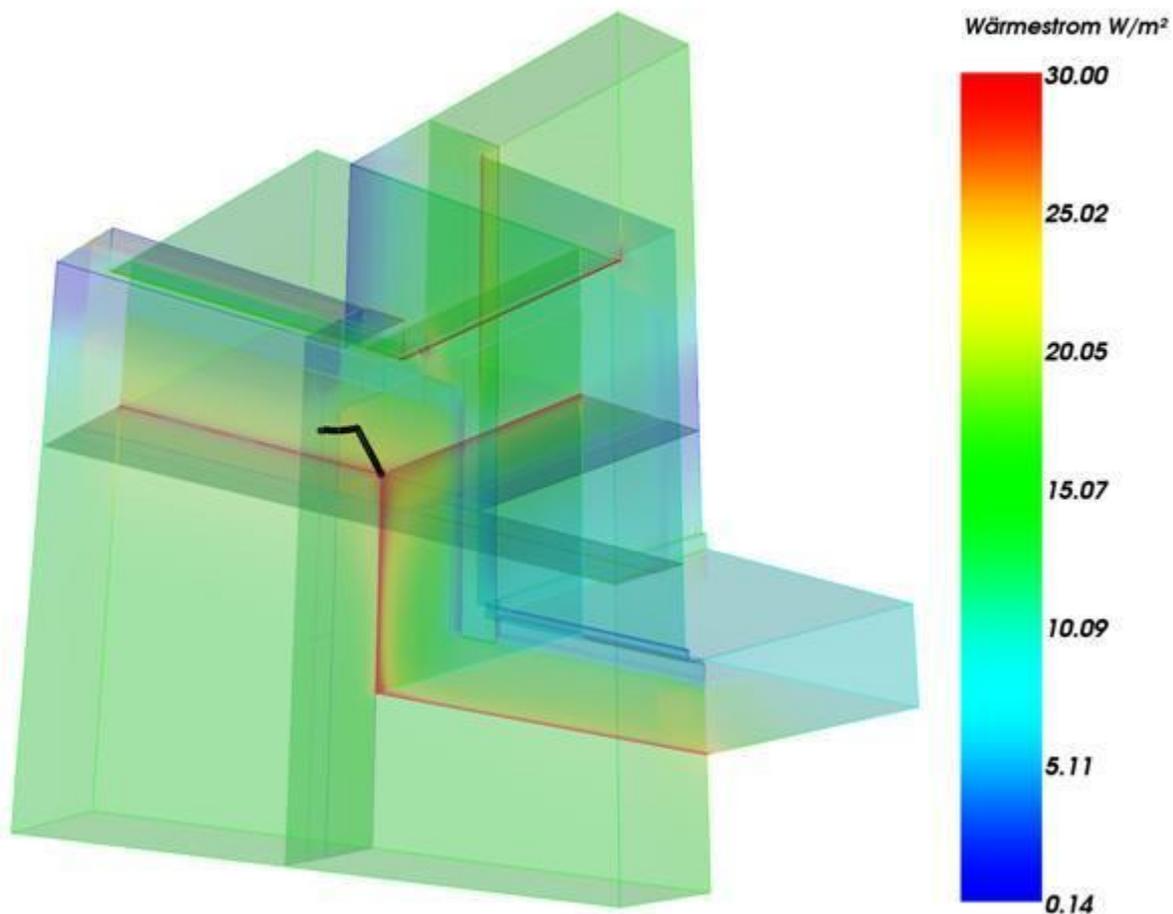


Abb. 3: Wärmestromlinie von der einspringenden Ecke nach außen

Es zeigt sich, dass die Wärme nicht den geometrisch kürzesten Weg nimmt, sondern in der Stahlbetondecke schräg nach oben fließt, der Dämmschicht des Flachdachs ausweicht und schließlich durch die Außenwand nach außen abfließt.

Als Planungsempfehlung ergibt sich nun unmittelbar, dass für eine Erhöhung der Temperatur in der einspringenden Ecke die Dämmung der Außenwand – und nicht jene des Flachdachs – verbessert werden muss.

4. Berechnung und Visualisierung der Diffusionsstromdichten

Bekanntlich kann der Transport von Wasserdampf durch eine Baukonstruktion mittels Diffusion mit einer formal der Wärmeleitungsgleichung entsprechenden Differentialgleichung beschrieben werden, sofern die Gültigkeit des 1. Fick'schen Gesetzes vorausgesetzt wird. Es ist daher nahe liegend, ein Wärmebrückenprogramm auch zur Beschreibung mehrdimensionaler ablaufender Wasserdampfdiffusionsprozesse heranzuziehen.

In AnTherm wurde diese Möglichkeit genutzt, sodass nach einer mehrdimensionalen Berechnung nicht nur das Temperaturfeld sondern auch das Feld des Wasserdampfpartialdrucks verfügbar ist.

Ein Vergleich der Verteilung des Wasserdampfpartialdrucks innerhalb der Baukonstruktion mit der aus den Temperaturen rückrechenbaren Verteilung des Sättigungsdampfdrucks zeigt unmittelbar, ob innerhalb des Bauteils mit Kondensatbildung gerechnet werden muss oder nicht.

Natürlich muss bei der Interpretation des Ergebnisses berücksichtigt werden, dass der Wasserdampftransport in Baukonstruktionen nicht allein durch Diffusion bewirkt wird und das Berechnungsergebnis somit nur als grobe Näherung für die in der Realität ablaufenden Vorgänge angesehen werden darf.

Dennoch zeigt sich, dass die zwei- oder dreidimensionale Beschreibung der Wasserdampfdiffusion in Baukonstruktionen sehr gut geeignet ist, klare Hinweise auf feuchtigkeitstechnische Schwachstellen der Konstruktion zu liefern und/oder wertvolle Hilfestellung bei der Suche nach der Ursache von Bauschäden aufgrund von Durchfeuchtung zu geben.

Dies soll im Folgenden am Beispiel des bereits behandelten, in [Abb. 1](#) dargestellten Bauteilausschnitts demonstriert werden. Zur Visualisierung des Diffusionsstroms werden die gleichen, von AnTherm angebotenen Methoden verwendet, wie für den Wärmestrom. Die folgende Abbildung zeigt ein Falschfarbenbild, mit dem die Verteilung der Beträge der Vektoren des Wasserdampfdiffusionsstroms an den Oberflächen und innerhalb der Schnittfläche dargestellt wird.

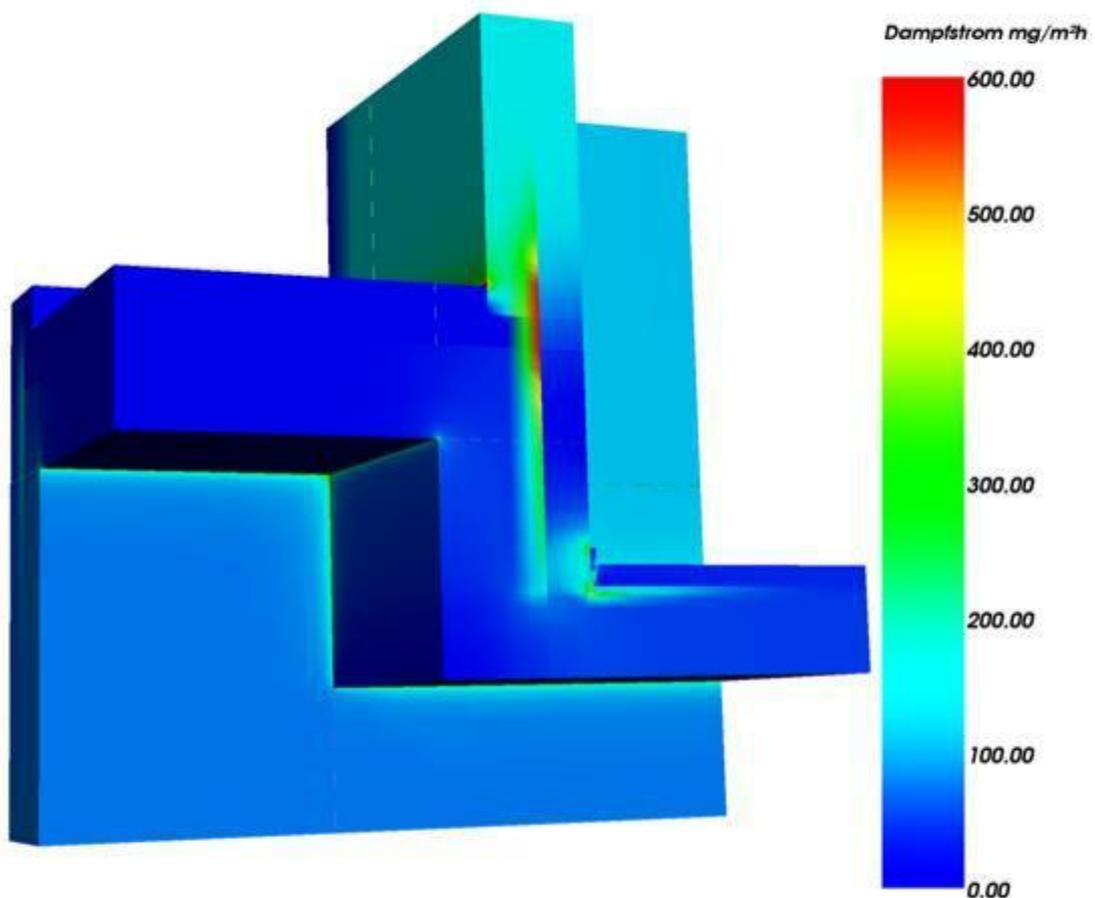


Abb. 4: Falschfarbenbild der Wasserdampfstromdichte

Der in Hinblick auf die Wasserdampfdiffusion kritische Bereich ist auf den ersten Blick erkennbar. Die mit Abstand höchste Dampfstromdichte stellt sich in der Kerndämmung der Wand zwischen Hallenbad und Wohnraum in Höhe des Flachdachs ein. Bedeutsam ist in diesem Zusammenhang, dass als Material der Kerndämmung Mineralfaserplatten, also ein Material mit extrem kleinem Diffusionswiderstand, verwendet wurde. Bautechnisch richtig wurde an der dem Wohnraum zugewandten Seite der Mineralfaserplatten eine PVC-Folie als Dampfbremse angebracht.

Der Weg des Dampfstroms durch die Konstruktion kann ähnlich wie beim Wärmestrom mittels Auszeichnen einer Stromlinie dreidimensional gezeigt werden. Es ist nahe liegend, sich durch Auswahl von Punkten innerhalb der

als kritisch erkannten Zone der Kerndämmung ein Bild über den Weg des Wasserdampfs durch die Konstruktion zu machen. Die folgende Abbildung zeigt als Beispiel eines der von AnTherm erzeugten Stromlinienbilder.

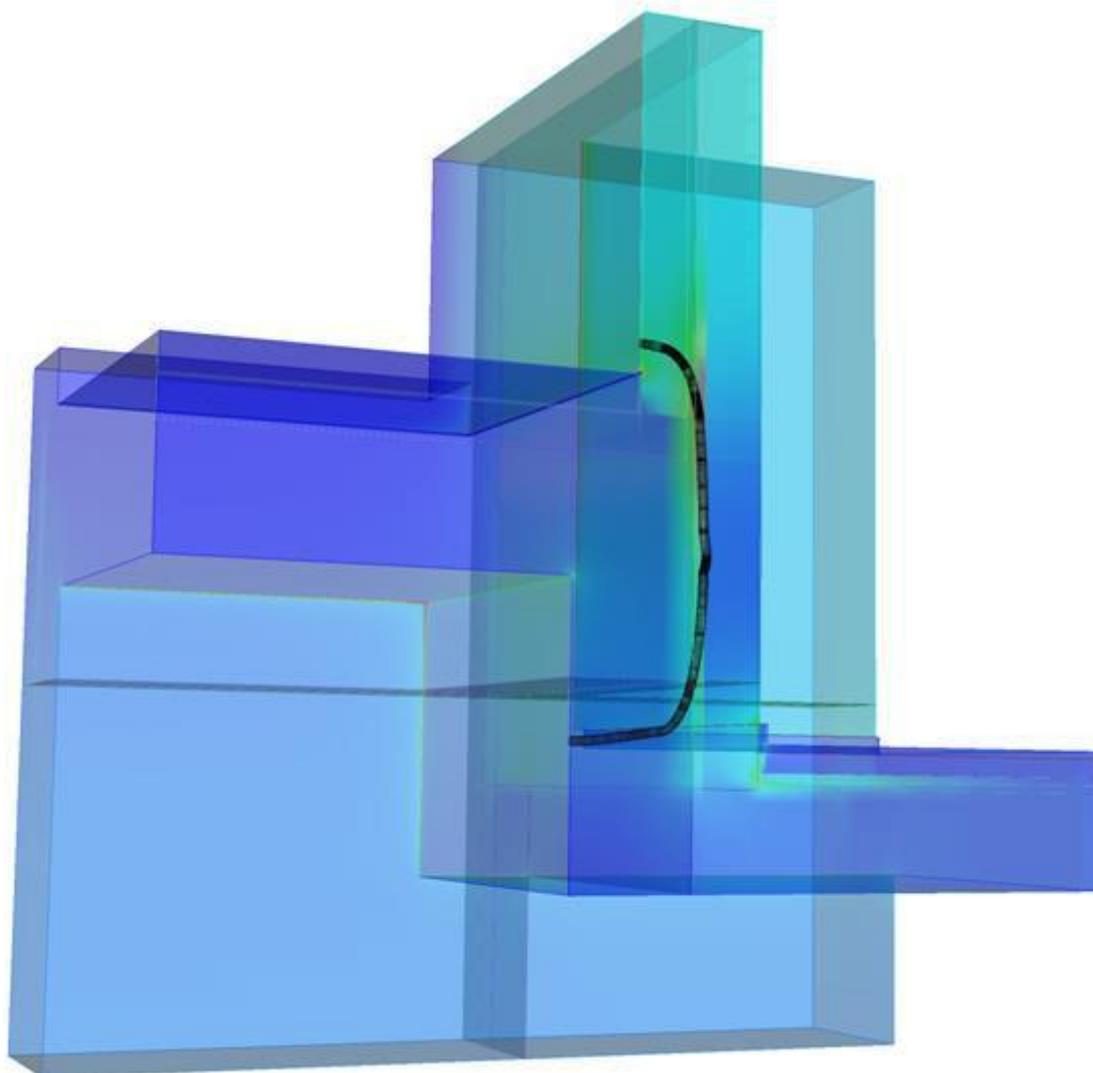


Abb. 5: Wasserdampfstromlinie von der inneren Oberfläche des Hallenbads nach außen

Es zeigt sich, dass der Wasserdampf vom Hallenbad den direktesten Weg in die Kerndämmung nimmt, dort aufgrund des kleinen Diffusionswiderstands weit nach oben steigt und schließlich durch die äußere Backstein-Schale nach außen abfließt. Da der berechnete Wasserdampfpartialdruck im Bereich dieser vorgemauerten Backsteinschale erheblich höher ist als der Sättigungsdampfdruck, muss davon ausgegangen werden, dass es dort zu massiver Kondensatbildung kommen wird.

Das gezeigte Beispiel erweist sich insofern als bedeutsam als in [\[2\]](#) berichtet wird, dass der dort zweidimensional betrachtete Fall einem gebauten Beispiel (Hallenbad in Poschavio in der Schweiz) entspricht, bei dem es zu starker Durchfeuchtung im Bereich der über dem Dach liegenden Außenwand der Wohnung gekommen ist.

Besonders interessant ist dabei, dass der normgemäß in eindimensionaler Näherung nach Glaser für die zweischalige Wand durchgeführten Nachweis (siehe z. B. ÖNorm B8110-2 [\[3\]](#)) auf keine Kondensatbildung führt, die Konstruktion also als vollkommen unkritisch einstuft. Dies zeigt, dass mit mehrdimensionalen Berechnungen nicht nur im Fall der Temperaturverteilung sondern auch im Fall der Verteilung des Wasserdampfpartialdrucks eine erhebliche Steigerung der Planungssicherheit erreicht werden kann.

5. Zusammenfassung

Die in Programmpaket AnTherm [1] umgesetzte Vorgangsweise zur Visualisierung der Verteilung von Wärme- und Wasserdampfdiffusionsströmen hat sich bereits als praxistaugliche Methode für das rasche Auffinden von thermischen und hygrischen Schwachstellen in Baukonstruktionen etabliert.

Das Visualisieren des Felds der Beträge der Wärme- und Diffusionsstromdichten mittels Falschfarbendarstellung zeigt auf einen Blick die kritischen Bereiche.

Die Möglichkeit, den Weg der Wärme und/oder des Wasserdampfdiffusionsstroms dreidimensional mittels Auszeichnen einer einzelnen Stromlinie zu verfolgen, gibt zudem wertvolle Hinweise in Hinblick auf Planungsempfehlungen zur Verbesserung der thermisch-hygrischen Qualität einer Baukonstruktion.

Literatur

- [1] Programmpaket AnTherm V1.0, Programm zur Analyse des Thermischen Verhaltens von Bauteilen mit Wärmebrücken und Dampfdiffusionsbrücken, © Tomasz Kornicki (2005)
- [2] Wieland, H. und Heim, U.: Mehrdimensionale Berechnung von Wärme- und Feuchtigkeitsströmen, ISBN 3-8169-0587-0, Expert-Verlag (1990)
- [3] ÖNorm B8110-2: Wärmeschutz im Hochbau; Teil 2: Wasserdampfdiffusion und Kondensationsschutz (2003)

Autoren

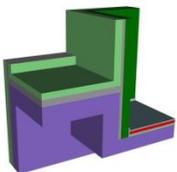
Ao. Univ. Prof. Dipl.-Ing. Dr. Klaus Kreč
Arbeitsgruppe für Nachhaltiges Bauen Institut
für Architektur und Entwerfen Technische
Universität Wien
A-1040 Wien, Karlsplatz 13/253.5
Österreich

Tomasz P. Kornicki Dienstleistungen
in EDV & IT A-1230 Wien,
Othello-gasse 1/RH8/2 Österreich

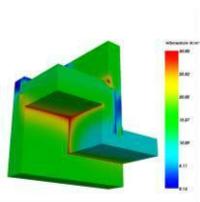
Abstract

Visualization of heat and water vapour diffusion flows in the case of tree-dimensional modelling; Klaus Kreč & Tomasz Kornicki

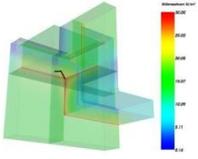
The difficulties in the depiction of the flow of heat and vapour through three-dimensionally modelled building constructions are taken up. A method well suited for practical use in finding regions which are critical with regard to heat and water vapour transport is presented. The pseudo colour image of the absolute values of the flow densities of heat and/or diffusion of water vapour immediately shows the regions of increased heat or vapour diffusion flow. Furthermore the plot of single flow lines clearly shows the three-dimensional path of the heat or water vapour through the building construction. The expressiveness of this method is demonstrated by an example representing a building construction which was analyzed by a three-dimensional calculation using the new thermal bridge program AnTherm.



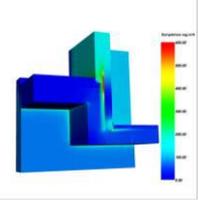
[Fig. 1:](#) three dimensional view of the building construction (model)



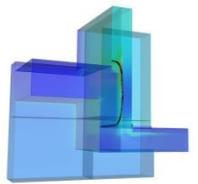
[Fig. 2:](#) pseudo colour image of the heat flow densities (heat flux)



[Fig. 3:](#) heat flow line depicting the path of heat through the building construction (heat stream line)



[Fig. 4:](#) pseudo colour image of the water vapour diffusion flow densities (vapour flux)



[Fig. 5:](#) heat flow line depicting the path of water vapour through the building construction (vapour stream line)

Kurzfassung

Die Problematik der Darstellung des Wärme- und/oder Dampfdiffusionsstroms durch dreidimensional modellierte Baukonstruktionen wird aufgegriffen und eine praxistaugliche Methode zur raschen Erkennung von thermischen und/oder hygrischen Schwachstellen vorgestellt. Das Falschfarbenbild der sich an den Oberflächen des untersuchten Bauteils einstellenden Beträge der Wärme- und/oder Dampfdiffusionsstromdichten zeigt unmittelbar Bereiche erhöhten Wärme- oder Diffusionsstroms auf. Das Auszeichnen einzelner Stromlinien gibt zudem Einblick in den Weg des Wärme- oder Diffusionsstroms durch die Baukonstruktion. Die Aussagekraft des vorgestellten Verfahrens wird anhand eines mit dem neuen Wärmebrückenprogramm AnTherm dreidimensional durchgerechneten Beispiels demonstriert.

Copyright

Dieses Artikel wurde im Juni 2006 im "[Bauphysik \(28\) Heft 3/2006](#)" auf Seiten 214-216 zuerst veröffentlicht (Ernst & Sohn Verlag für Architektur und technische Wissenschaften GmbH & Co. KG, Berlin). Nach Ablauf der dreijährigen Sperrfrist haben die Autoren seit Juli 2009 das Recht anderweitigen Wiederveröffentlichungen.

Der Abstract des Artikels (© 2006 WILEY-VCH Verlag GmbH & Co. KGaA, Weinheim) wurde mit der Einwilligung von Ernst & Sohn Verlag für Architektur und technische Wissenschaften GmbH & Co. KG, Berlin übernommen.

The building envelope as thermal heat bridge

This essay is available in German only.

Die Gebäudehülle als Wärmebrücke

Die Transmissionswärmeverluste der Gebäudehülle beeinflussen die im Inneren des Gebäudes zur Einhaltung einer Soll-Temperatur erforderliche Heizleistung nachhaltig. Die Planung von Gebäuden mit niedrigem Heizwärmebedarf ist immer mit der Forderung einer Optimierung der Gebäudehülle in Hinblick auf die Reduktion der zu erwartenden Transmissionswärmeverluste verknüpft. Dies gilt auch für „sonnengestützte Häuser“, also für Gebäudeentwürfe, deren primäres Planungsziel eine möglichst gute Ausnutzung der Energie der auf das Gebäude einfallenden Sonnenstrahlung zwecks Reduzierung der erforderlichen Heizenergie ist.

Um die thermischen Eigenschaften der Gebäudehülle erfolgreich optimieren zu können, bedarf es planungsbegleitender Berechnungen. Die hierzu verwendeten Hilfsmittel sind Computerprogramme, die es gestatten sowohl das thermische Verhalten einzelner Teile der Gebäudehülle als auch das thermische Verhalten des gesamten Gebäudes in Abhängigkeit von der Planungsvariante vorher zu sagen. Für die Entwickler solcher Programme sind die Anforderungen insofern besonders hoch, als einerseits die physikalische Modellierung detailliert genug sein muss, um Ergebnisse in solch' einem Grad an Genauigkeit zu erhalten, dass sie dem Anspruch an eine hohe Planungssicherheit gerecht werden können. Andererseits muss bedacht werden, daß Überlegungen in Hinblick auf die Optimierung der thermischen Qualität des Gebäudes in frühen Planungsphasen besonders effektiv sind. Zielgruppe für Programme zur Beurteilung der thermischen Qualität von Baukonstruktionen und Gebäuden sollte aus diesem Grunde in erster Linie die Gruppe der PlanerInnen selbst sein und erst in zweiter Linie jene der Spezialfachleute wie z. B. BauphysikerInnen oder HaustechnikerInnen.

Die Beschreibung des Transmissionswärmeverlustes eines Bauteils ist nur bei Verwendung eines thermisch eindimensionalen Modells der Handrechnung zugänglich. Die Erfahrung mit diesen einfachen Berechnungsmethoden zeigte bei nicht oder nur wenig wärmedämmten Gebäuden eine erstaunlich gute Übereinstimmung zwischen berechnetem „Bedarfs-Wert“ und gemessenen „Verbrauchs-Werten“, sofern bei der Flächenermittlung auf Außenmaße zurückgegriffen wurde. Mit steigendem Anspruch an die thermische Qualität des Gebäudes wird jedoch das Rechnen mit eindimensionalen Modellen zunehmend ungenauer. Grund für diesen Trend sind mehrdimensional ablaufende Wärmetransportvorgänge, deren Auswirkung auf die Transmissionswärmeverluste der gesamten Gebäudehülle mit besser werdender Wärmedämmung stark ansteigt. Bei Gebäuden, deren Wärmedämmung den heute gültigen Bauordnungen und Verordnungen entspricht, ist die eindimensionale Rechnung bereits so ungenau, daß die erforderliche Planungssicherheit nicht mehr gegeben ist. Bei Passivhäusern mit ihren extrem hohen Ansprüchen an die Planungssicherheit ist das Erfassen von mehrdimensional ablaufenden Wärmeleitungsvorgängen – eine Thematik, die gerne unter dem Schlagwort „Wärmebrücken“ zusammengefasst wird – zu einem zentralen Punkt der Bauplanung geworden.

Das Erfassen des thermischen Verhaltens von Wärmebrücken ist bei allen für die Baupraxis relevanten Fragestellungen nur mit dem Einsatz numerischer Methoden unter Zugriff auf so genannten „Wärmebrücken-Programme“ möglich. Obzwar mittlerweile eine Vielzahl solcher Programme existiert und am Markt angeboten wird, besteht die Problematik, daß die Handhabung der meisten dieser Programme für den Planer zu aufwendig ist. Aus diesem Grund werden Wärmebrückenberechnungen nach wie vor von Spezialfachleuten oder aber überhaupt nicht durchgeführt. Auch auf dem Gebiet der Planung von Passivhäusern findet eine planungsbegleitend durchgeführte rechnerische Optimierung der thermischen Eigenschaften von Bauteilanschlüssen nicht generell statt. Die Praxis, auf „passivhaustaugliche“ Komponenten zuzugreifen, bringt zwar die erforderliche Planungssicherheit mit sich. Die wünschenswerte Förderung innovativer Planungslösungen wird auf diese Weise jedoch nicht unterstützt.

Im Folgenden wird zunächst der wissenschaftliche Hintergrund und die normative Lage in Hinblick auf die Erfassung mehrdimensionaler Wärmeleitungsvorgänge kurz dargelegt und der derzeitige Stand der Technik bezüglich der angebotenen Wärmebrückenprogramme analysiert. Schließlich wird die Richtung einer gleichermaßen wünschenswerten wie auch notwendigen Weiterentwicklung auf dem Gebiet der thermischen Modellierung skizziert.

Lesen Sie hier weiter:

- [Die Gebäudehülle als Wärmebrücke](#) 

The above essay has been incorporated with friendly permission from Klaus Kreč, Büro für Bauphysik, A-3562 Schönberg am Kamp, Veltlinerstr. 9, Österreich/Austria .

See also: [Theoretical background](#)

On the Storage of Heat in Building Components

Examining heat storage in building construction materials and assemblies - often referred to as thermal storage - by means of a periodically oscillating model of thermal conduction leads to a generalized conductance concept (complex conductance matrices). This concept can serve as the basis for three- dimensional, non-steady-state (transient) calculations of the thermal performance of buildings. Relationships to the concept of effective thermal capacity (previously introduced in other publications) are established in context.

...

A correct treatment of heat storage processes in building structures is only possible by means of a thorough examination of relevant time-dependant heat flows. The concept presented here is only seemingly restricted to periodically fluctuating processes since a generalization which incorporates aperiodic functions as well is principally possible with the use of Laplace transforms.

The segment of this discourse most relevant to the practical application of evaluating a building's thermal performance is the generalization of the already familiar conduction value concept for steady-state conditions to include periodically time-dependant conditions as well. This provides the basis for calculating the thermal performance of entire buildings in which two- and three-dimensional heat conduction is considered - without a serious increase in complications as compared to one-dimensional, time-dependant calculations.

Continue reading here:

- [On the Storage of Heat in Building Components \(pdf\)](#) 

The above essay has been incorporated with friendly permission from *Klaus Kreč, Büro für Bauphysik, A-3562 Schönberg am Kamp, Veltlinerstr. 9, Österreich/Austria* .

See also: [Theoretical background](#)

Transient, multidimensional calculations by an example of components in contact with soil



This essay is available in German only.

Instationäre, mehrdimensionale Berechnungen am Beispiel erdbodenberührter Bauteile



Die Instationär-Version von AnTherm bietet die Möglichkeit, im Zuge mehrdimensionaler Berechnungen auch die Einflüsse der Wärmespeicherfähigkeit zu berücksichtigen. Nun mag man vielleicht meinen, dass die Wärmespeicherfähigkeit nur im Sommer eine Rolle spielt und ihr in Zusammenhang mit Wärmebrückenberechnungen bestenfalls eine akademische Bedeutung zukommt. Diese Meinung mag in Zusammenhang mit luftberührten Bauteilanschlüssen durchaus berechtigt sein, sofern nur der Wärmedurchgang interessiert. Schon bei der Frage nach Oberflächentemperaturen oder Temperaturen im Bauteil wird die instationäre Berechnung bei massiver Bauweise zu deutlich anderen Planungsempfehlungen und Bewertungen führen als eine stationäre oder quasistationäre Rechnung.

Bei einem wichtigen Sonderfall spielt die Wärmespeicherfähigkeit allerdings eine derart dominante Rolle, dass die Sinnhaftigkeit einer stationären oder quasistationären Betrachtungsweise von vornherein in Frage gestellt werden muss. Im Fall erdbodenberührter Bauteile sind weite Bereiche des ein Gebäude umgebenden Erdreichs in den Wärmetransport vom Gebäudeinneren nach außen einbezogen, sodass allein aufgrund der extrem großen Masse des vom Wärmedurchgang betroffenen Erdreichs Wärmespeichereffekte keinesfalls vernachlässigt werden können.

Die Auswirkung der Vernachlässigung des Wärmespeichervermögens kann mit einem Blick auf das sommerliche Raumverhalten rasch plausibel gemacht werden: je größer das Wärmespeichervermögen der Bauteile ist, desto kleiner wird die Tagesschwankung der Innenlufttemperatur und in der Folge auch die Schwankung der Wärmeströme. Bei bodenberührten Bauteilen wirkt sich die Tagesschwankung der Außenlufttemperatur auf das Temperaturfeld im Erdboden kaum aus; aufgrund der extrem hohen Wärmespeicherfähigkeit spielt hier nur die Jahresschwankung der Außenlufttemperatur eine Rolle. Aber auch hier gilt: je größer die Wärmespeicherfähigkeit, desto kleiner ist die Jahresschwankung der Wärmeverluste. Wird nun der Einfluss der Wärmespeicherfähigkeit ignoriert, wie das bei allen stationären und quasistationären Berechnungsansätzen der Fall ist, erhält man eine viel zu große Jahresschwankung der Wärmeverluste. Das viel zu hohe Maximum der Wärmeverluste liegt dabei im Tiefwinter, also genau in jener Zeit, in dem eine möglichst genaue Erfassung der Wärmeverluste besonders wichtig ist. Bezogen auf die Normverfahren bedeutet dies, dass die quasistationäre, monatliche Berechnung mit Temperaturkorrekturfaktoren („f-Werten“), wie sie z. B. in der ÖNorm B8110-6 [1] oder in der Energieausweis-Berechnung verankert ist, tendenziell auf viel zu hohe Wärmeverluste führt.

Zwar liegen diese quasistationären Ansätze auf „der sicheren Seite“, doch liegt die Größenordnung dieser Sicherheit in Bereichen, wo deren Sinnhaftigkeit angezweifelt werden muss [2]. Insbesondere bei der Planung von Niedrigenergiegebäuden, Passivhäusern und Plusenergiehäusern kann der Fehler einer nur überschlägigen (, doch sehr wohl normgemäßen) Berechnung der Wärmeverluste über die erdbodenberührten Bauteile in der gleichen Größenordnung liegen, wie die Summe der Wärmebrückenzuschläge für alle Bauteilanschlüsse des Gebäudes. Bei Gebäuden mit großflächigen bodenberührten Bauteilen, wie z. B. bei Werks- oder Lagerhallen, ist der Wärmeabflusses über den Erdboden nach außen und zum Grundwasser zumeist der bestimmende Faktor für das thermische Gebäudeverhalten. Bei einer zu ungenauen Erfassung dieser Wärmeverluste liegt die Gefahr von Fehleinschätzungen und Fehlplanungen sehr nahe.

Auf dem Gebiet der Wärmeverluste über erdbodenberührte Bauteile besteht offenbar dringender Bedarf nach verfeinerten Berechnungsmethoden. Diese sollen helfen, die Planungssicherheit zu erhöhen und die Bewertung der thermisch-energetischen Qualität von Gebäuden realitätsnäher zu gestalten. Dies war ein Grund für die Entwicklung eines Anwenderbezogenen Simulationsprogramms, mit dem die Jahresverläufe der Wärmeverluste über erdbodenberührte Bauteile mit bisher nicht gekannter Genauigkeit berechnet werden können.

Es war insofern nicht weiter schwierig, AnTherm und THESIM miteinander zu verknüpfen, als beiden Programmen der gleiche theoretische Ansatz zugrunde liegt. In der auf instationäre, periodisch eingeschwungene Vorgänge verallgemeinerten Leitwert-Theorie kann für jede Harmonische (d. h. für jeden rein sinusförmigen Zeitverlauf) die Wärmeleitungsgleichung exakt gelöst werden [3]. Da der aus den langjährigen Monatsmittelwerten gebildete, geglättete Jahresgang der Außenlufttemperatur sehr gut mit einer reinen Sinuskurve approximiert werden kann, bringt bereits die Rechnung mit nur einer Harmonischen eine brauchbare Näherung; durch Hinzunahme mehrerer Harmonischer kann eine Abweichung vom rein sinusförmigen Verlauf beliebig gut modelliert und die Genauigkeit des Berechnungsergebnisses nach Belieben gesteigert werden.

Die instationäre Leitwert-Theorie [3] kann als Grundlage eines Gebäudesimulationsprogramms, das die Mehrdimensionalität von Wärmedurchgang und Wärmespeicherung zu erfassen gestattet, verwendet werden. Programmpaket THESIM wurde nach den in [4] beschriebenen, auf die Leitwert-Theorie aufbauenden Simulationsalgorithmen entwickelt. Für den hier interessierenden Spezialfall des Wärmedurchgangs durch den Erdboden wurde das Simulationsprogramm THESIM mit einer Benutzeroberfläche kombiniert, die zwar die vielfältigen

Möglichkeiten des Simulationsprogramms nur eingeschränkt nutzt, die Handhabung für den Benutzer damit jedoch sehr einfach werden lässt. Auf diese Weise wird erreicht, dass selbst dreidimensionale, instationäre Simulationen ohne großen Aufwand und ohne weitreichendes Spezialwissen möglich werden.

Als pdf lesen: [Instationäre, mehrdimensionale Berechnungen am Beispiel erdbodenberührter Bauteile](#)



Die obigen Artikel wurden mit der freundlichen Einwilligung von *Klaus Kreč, Büro für Bauphysik, A-3562 Schönberg am Kamp, Veltlinerstr. 9, Österreich* übernommen.

[1] ÖNorm B8110-6, Wärmeschutz im Hochbau – Teil 6: Grundlagen und Nachweisverfahren – Heizwärmebedarf und Kühlbedarf, Ausgabe 2011-03-01

[2] J. N. Nackler, Wärmeverluste erdbodenberührter Bauteile im Passivhausstandard – Vergleich verschiedener Berechnungsverfahren, Masterarbeit, Fakultät für Architektur und Raumplanung der TU Wien, 2010 (als Download unter www.thesim.at verfügbar)

[3] [K. Kreč, Zur Wärmespeicherung in Baukonstruktionen, Gesundheits-Ingenieur 114, Heft 1, 11-18, 1993](#) (als Download unter www.thesim.at verfügbar)

[4] [K. Kreč, Zur dreidimensionalen Simulation des thermischen Verhaltens von Gebäuden, Gesundheits-Ingenieur 121, Heft 6, 293-344, 2000](#) (als Download unter www.thesim.at verfügbar)

Siehe auch: [Theoretische Grundlagen](#)

Heat Transfer in Building Constructions in consideration of Heat Sources

This essay is available in German only.

Wärmeleitung in Baukonstruktionen unter Berücksichtigung von Wärmequellen

Zum Thema der Wärmeleitern in Baukonstruktionen existiert eine umfangreiche Literatur. In der Mehrzahl der Fälle wird dabei nur der eindimensionale Fall der stationären Wärmeleitung abgehandelt. Der in der Realität vorliegende Fall der dreidimensionalen instationären Wärmeleitung wurde der praktischen Berechnung erst in letzter Zeit unter Zuhilfenahme der elektronischen Datenverarbeitung zugänglich. Ein derartiges Konzept, das zumindest zum Teil schon in anwendbare EDV-Programme umgesetzt ist, wird z. B. in [1] (stationärer Fall) und [2] (instationärer Fall) geschildert.

Bedenkt man, dass die thermischen Vorgänge in einem Bauwerk nicht nur durch die Außenlufttemperatur und die Lufttemperaturen in seinen Innenräumen bestimmt werden, sondern ebenso durch die Wärmequellen, die infolge von Sonneneinstrahlung, langwelliger Wärmestrahlung und natürlich auch beim Betrieb von Heizkörpern auftreten, so wird einem klar, dass jedes Konzept, das diese Wärmequellen nicht berücksichtigt, unvollständig und wirklichkeitsfremd bleiben muss.

...

Wie solche Wärmequellen berücksichtigt werden können, wurde schon 1975 unter Beschränkung auf eindimensionale Wärmeleitung und flächenhafte Wärmequellen in [4] für den periodisch eingeschwungenen Fall besprochen. Hier soll der dreidimensionale Fall und der Zusammenhang mit dem in [2] geschilderten Leitwert-Konzept behandelt werden.

...

Die Berücksichtigung von Wärmequellen innerhalb oder an den Oberflächen einer Baukonstruktion ist im Rahmen des in [2] vorgestellten Leitwert-Konzeptes sowohl im stationären als auch im periodisch eingeschwungenen Fall ohne weiteres möglich. Die Auswirkung von Wärmequellen auf die thermischen Vorgänge im Bauwerk lassen sich unabhängig von den jeweiligen Randbedingungen und unabhängig vom Wert der den Wärmequellen zugeordneten Heizleistungen beschreiben. Als kennzeichnende Größen zur Charakterisierung einer Wärmequelle dienen im Rahmen des hier vorgestellten Konzeptes die Verteilungsschlüssel und die Heizleistungsamplituden.

Die Berechnung von Heizleistungs-Verteilungsschlüsseln und von Leitwert-Matrizen (siehe [2]) ermöglicht es, das thermische Verhalten von Baukonstruktionen unter Berücksichtigung zwei- und dreidimensionaler Wärmeleitungsvorgänge instationär zu erfassen. Insbesondere liefert sie die grundlegenden Daten für dreidimensional rechnende Programme zur Simulation des thermischen Verhaltens ganzer Gebäude unter periodisch eingeschwungenen Verhältnissen.

Lesen Sie hier weiter:

[Wärmeleitung in Baukonstruktionen unter Berücksichtigung von Wärmequellen \(pdf\)](#) 

The above essay has been incorporated with friendly permission from *Klaus Kreč, Büro für Bauphysik, A-3562 Schönberg am Kamp, Veltlinerstr. 9, Österreich/Austria*.

[1] Heindl, W., Kreč K., Panzhauser E. und Sigmund A.: *Wärmebrücken*. Springer-Verlag Wien-New York 1987.

[2] Kreč, K.: *On the Storage of Heat in Building Components (Zur Wärmespeicherung in Baukonstruktionen)*. *Gesundheits-Ingenieur* 114 (1993), H. 1, S. 11-18.

[3] Koch, H. A. und Pechinger, U.: Möglichkeiten zur Berücksichtigung von Sonnen- und Wärmestrahlungseinflüssen auf Gebäudeoberflächen. *Gesundheits-Ingenieur* 98(1977), H. 10, S. 265—280.

[4] Haferland, Fr., Heindl, W und Fuchs, H.: Ein Verfahren zur Ermittlung des wärmetechnischen Verhaltens ganzer Gebäude unter periodisch wechselnder Wärmeeinwirkung und rechnerische Untersuchungen zur Ermittlung der Größenordnung bestimmter Einflüsse von Bauweise und Konstruktion sowie sonstiger Parameter auf die Temperaturstabilität in Räumen. *Berichte aus der Bauforschung* H. 99, Verlag Wilhelm Ernst & Sohn Berlin — München — Düsseldorf 1975.

[5] Frank, P. und Mises, R.: *Die Differential- und Integralgleichungen*. Band 2. Friedr. Vieweg & Sohn, Braunschweig 1961.

See also: [Theoretical background](#)

A practical method for the assessment of heat bridges with a view to surface condensation

A practical method is presented to be used for assessing the steady-state thermal behaviour of heat bridges. This method bases on the fact, that the temperature at each point along the heat bridge can be represented as linear combination of the adjacent air temperatures. A single computer calculation of basic solutions allows to determine the standardized weights of these air temperatures for the construction concerned. Such calculations can be performed on personal computers without any difficulties by means of a specialized programme package. Both, this fact and the simple possibility of the subsequent determination of surface temperatures under arbitrary boundary conditions support the hope that heat bridges will no longer frighten us in future.

This essay is available in German only.

Ein praktisches Verfahren zur Beurteilung von Wärmebrücken im Hinblick auf Oberflächenkondensation

Ein praktisches Verfahren zur Beurteilung des stationären thermischen Verhaltens von Wärmebrücken wird vorgestellt. Es beruht auf der Tatsache, dass sich die Temperatur an jeder Stelle der Wärmebrücke als Linearkombination der angrenzenden Lufttemperaturen darstellen lässt. Die normierten Gewichte dieser Lufttemperaturen können durch eine einmalige EDV-Berechnung von Basislösungen für die jeweils vorliegende Konstruktion ermittelt werden. Solche Berechnungen sind mittels eines eigens entwickelten Programmpaketes auf Personal-Computern problemlos durchführbar. Dies und die einfache Möglichkeit der anschließenden Ermittlung von Oberflächentemperaturen unter beliebigen Randbedingungen geben zur Hoffnung Anlass, dass Wärmebrücken in Zukunft ihren Schrecken verlieren.

...

Die Autoren haben in Zusammenarbeit mit W. Heindl [4] Verfahren ... entwickelt, die — einmal angewendet — die thermische Beurteilung von Wärmebrücken im stationären Fall unter den verschiedensten Randbedingungen auf ähnlich einfache Weise ermöglichen, wie bei „eindimensional“ erfassbaren Bauteilen. Das Verfahren beruht auf der Tatsache, dass die Fouriersche Wärmeleitungsgleichung bei Abwesenheit von Wärmequellen linear und homogen ist, jede Superposition von Lösungen daher wieder Lösung der Differentialgleichung ist. Es bedarf daher nur geschickt gewählter, linear unabhängiger „Basislösungen“, um die zu bestimmten Randbedingungen gehörige spezielle Lösung durch Superposition — also ohne weitere aufwendige Berechnung — zu erhalten.

In diesem Aufsatz wollen wir uns auf das Problem der Bestimmung der Oberflächentemperaturen im Wärmebrückenbereich beschränken. Das Problem der Bestimmung der Wärmeströme zwischen den durch eine Wärmebrücke thermisch verbundenen Räumen sei einer späteren Veröffentlichung vorbehalten.

...

Das bestehende Programmpaket zur Berechnung der „Gewichte“ an kritischen Stellen von Wärmebrücken und die Einfachheit der Verwendung dieser — einmal zu berechnenden — Gewichte zur Bestimmung der Oberflächentemperaturen sollten uns ermutigen, die nötigen Untersuchungen von Wärmebrücken nicht erst bei bereits ins Auge springenden Bauschäden zu beginnen.

Hat man sich einmal vor Augen geführt, dass die Temperatur an jeder Stelle einer Wärmebrücke sich als Linearkombination der Lufttemperaturen der angrenzenden Räume ergibt, so ist nichts naheliegender, als die Gewichte (g -Werte) an den interessierenden Stellen des Wärmebrückenbereiches zu ermitteln; dies erfordert zwar einen einmaligen nennenswerten Aufwand, ermöglicht aber anschließend problemlose Berechnungen der Temperaturen an diesen Stellen, wobei nun der Aufwand kaum größer ist als bei Temperaturberechnungen im eindimensionalen Fall.

Allen Betroffenen — vom Planer bis zum Benutzer des Gebäudes — wäre gedient, wenn aufgrund entsprechender normativer Festlegungen der Nachweis zu erbringen wäre, dass die verwendete Wärmebrückenkonstruktion bestimmten thermischen Anforderungen genügt. Bei ebenen Wänden werden derartige Nachweise schon lange verlangt (Mindestwärmeschutz); bei Wärmebrücken wären sie ebenfalls ohne weiteres möglich.

Solche Normen sollten vom Planer verlangen, die Gewichte an kritischen Stellen der Konstruktion auszuweisen. Auf dieser Basis lassen sich für beliebige Lufttemperaturen der angrenzenden Räume die zugehörigen Oberflächentemperaturen leicht berechnen; damit können bereits quantitative Aussagen über die Gefahr der Bildung von Oberflächenkondensat gemacht werden.

Die rasche Verbreitung leistungsfähiger Personal-Computer lässt erwarten, dass Rechenprogramme für stationäre mehrdimensionale Wärmeleitungsprobleme bald allgemein zugänglich und routinemäßig angewendet werden können. Alle Versuche, Wärmebrücken zu typisieren und deren thermisches Verhalten durch eine Vielzahl empirischer Formeln zu charakterisieren, erübrigen sich damit.

Lesen Sie hier weiter:

- [Ein praktisches Verfahren zur Beurteilung von Wärmebrücken im Hinblick auf Oberflächenkondensation \(pdf\)](#)



The above essay has been incorporated with friendly permission from *Klaus Kreč, Büro für Bauphysik, A-3562 Schönberg am Kamp, Veltlinerstr. 9, Österreich/Austria*.

[1] Mainka, G.-W., Paschen, H.: Wärmebrückenkatalog. B.G. Teubner, Stuttgart 1986.

[2] Erhorn, H., Tammes, E.: Eine einfache Methode zum Abschätzen balkenförmiger Wärmebrücken in Bauteilen mit planparallelen Oberflächen. Bauphysik (1985), H. 1, S. 7—11.

[3] Berber, J.: Außenwinkel als Wärmebrücken. Bauphysik 6 (1984), H. 4, S. 142—144.

[4] [Heindl, W., Kreč K., Panzhauser E. und Sigmund A.: Wärmebrücken. Springer-Verlag Wien-New York 1987.](#)

See also: [Theoretical background](#)

The "super" - refinement

The gridding (discretization) algorithm of AnTherm subdivides the component model under consideration into cells which will be thermally balanced thereafter. The calculation algorithm creates for each such cell the heat balance equation and then solves the system of such equations. The solution results describe temperature field with values in the middles of cells.

During the evaluation AnTherm will provide further *refinement* of this results. Here, by using *heat balance and interpolation algorithms*, AnTherm uses temperatures at cell centres to calculate temperatures at centres of cell edges, centres of cell surfaces and at each cell corner.

As the nice result of such refinement the provisioning of much higher number of node values is available for the evaluation, hence the evaluation can be much more precise thereafter - for the two dimensional case the number of temperature nodes is fourfold the original cell count and in three dimensional case it is eightfold!

For example, the calculation with 1.000.000 (one million) balanced cells provides results of thermal balance equations **at more than 8.000.000 (eight million) points!** - and the operation is accomplished in surprisingly short computation time. In most cases in less then one minute!

The most beneficial effect of this *refinement* can be seen in that the *temperatures at any material boundary* can be provisioned that way! Material boundaries of special interest exist at the junction of different materials and, more importantly, at all surfaces of the component considered for analysis.

Evaluating temperatures at component surfaces is essential to the evaluation and analysis of thermal bridges. Therefore the *super refinement* shall be seen as substantial calculation step.



Appropriate responses to the complex problem of finding the cause for mould growth?

This essay is available in German only.

Angemessene Antworten auf das komplexe Problem der Schimmelursachen?

Die DIN 1946-6 spricht von der "Lüftung zum Feuchteschutz ohne Nutzerunterstützung" - genauere Analyse der unvermeidbaren Feuchtlast - zur Vermeidung von Schimmel an den wärmeschutztechnisch ungünstigsten Stellen der nutzerunabhängige Grundluftwechsel nur durch bauliche Maßnahmen (mindestens passiv) sichergestellt werden kann.

Im September 2010 ist der DIN - Fachbericht 4108-8 mit dem Titel: "Wärmeschutz und Energieeinsparung in Gebäuden, Teil 8: Vermeidung von Schimmelwachstum in Wohngebäuden" erschienen.

Im Gegensatz zu Normfassungen wurde der Fachbericht nicht als Entwurfsfassung vorgelegt, so dass die breitere Fachöffentlichkeit keine Möglichkeit zur Mitwirkungs- und Einspruchsmöglichkeit hatte.

Viele Inhalte des Fachberichts sind zu begrüßen, andere sind kritikwürdig oder sogar unzutreffend.

Prof. Dr.-Ing. R. Oswald hat dazu eine Stellungnahme verfasst, in der auf eine schnelle Korrektur des Fachberichts gedrängt wird.

Die Stellungnahme ist in der Zeitschrift „Der Bausachverständige“, Ausgabe 1/2011, erschienen.

Lesen Sie hier weiter:

- ["Angemessene Antworten auf das komplexe Problem der Schimmelursachen?"
Ein Sonderdruck und Stellungnahme Stellungnahme zum DIN-Fachbericht 4108-8 Vermeidung von Schimmelwachstum in Wohngebäuden \(pdf\)](#) 

Siehe auch: [Theoretische Grundlagen](#)

The concept of thermal conductance

This essay is available in German only.

Das Leitwert-Konzept

Eine wichtige Grundlage für die Beschreibung des thermischen Verhaltens eines Körpers stellt der Fourier'sche Wärmestromansatz ...

Lesen Sie hier weiter:

- [Das Leitwert-Konzept \(pdf\)](#)



The above essay have been incorporated with friendly permission from *Klaus Kreč, Büro für Bauphysik, A-3562 Schönberg am Kamp, Veltlinerstr. 9, Österreich/Austria* .

- [1] Kreč K.: Bewertung der Sommertauglichkeit von Gebäuden, Forschungsbericht im Auftrag der Initiative Ziegel im Fachverband der Stein- und keramischen Industrie, 31. März 2006
- [2] Stockinger, J., Kreč, K. & Seidl, J.: Thermisch optimierter Einsatz von Ziegel in Passivhäusern, Tagungsbericht der 8. Europäischen Passivhaustagung 2004, 205-219, Krems (2004)
- [3] Stockinger, J.: Thermisch optimierter Einsatz von Ziegel im Passivhaus, Master-Thesis, Donau-Universität Krems (2004)
- [4] Bruckmayer, F.: Der praktische Wärme- und Schallschutz im Hochbau, Deuticke-Verlag Wien (1949)
- [5] Koch H. A. & Pechinger, U.: Möglichkeiten zur Berücksichtigung von Sonnen- und Wärmestrahlungseinflüssen auf Gebäudeoberflächen. Gesundheits-Ingenieur 98, H. 10, S. 265-280, (1977)
- [6] [Heindl W., Kreč K., Panzhauser, E. & Sigmund, A.: Wärmebrücken. Springer-Verlag Wien-New York \(1987\)](#)
- [7] ÖNorm B8110-1, Wärmeschutz im Hochbau – Anforderungen an den Wärmeschutz und Nachweisverfahren. (2000)
- [8] EN ISO 10211-1: Wärmebrücken im Hochbau – Wärmeströme und Oberflächentemperaturen – Teil 1: Allgemeine Berechnungsverfahren. (1996)
- [9] [Kreč, K.: Zur Wärmespeicherung in Baukonstruktionen. Gesundheits-Ingenieur 114, H. 1, S. 11-18, \(1993\)](#)
- [10] ÖNorm EN ISO 13786: Wärmetechnisches Verhalten von Bauteilen – Dynamisch-thermische Kenngrößen – Berechnungsverfahren, Ausgabe: 2000-08-01 (2000)
- [11] [Kreč, K.: Zur dreidimensionalen Simulation des thermischen Verhaltens von Gebäuden“, Gesundheits-Ingenieur 121, Heft 6, 293-344 \(2000\)](#)
- [12] ÖNorm B8110-3, Wärmeschutz im Hochbau – Wärmespeicherung und Sonneneinflüsse, Ausgabe: 1999-12-01 (1999)
- [13] EN 832, Wärmetechnisches Verhalten von Gebäuden – Berechnung des Heizenergiebedarfs – Wohngebäude (1998)
- [14] ÖNorm EN ISO 13791, Wärmetechnisches Verhalten von Gebäuden - Sommerliche Raumtemperaturen bei Gebäuden ohne Anlagentechnik - Allgemeine Kriterien und Validierungsverfahren (2004)
- [15] ÖNorm EN ISO 6946, Bauteile – Wärmedurchlasswiderstand und Wärmedurchgangskoeffizient – Berechnungsverfahren (2005)
- [16] Minke, G.: Lehm- und Ökobuch, Staufen bei Freiburg (1999)
- [17] [AnTherm V3.51, Programmpaket zur Analyse des Thermischen Verhaltens von Bauteilen mit Wärmebrücken, © 2004 – 2007 T. Kornicki \(2007\)](#)
- [18] GEBA V7.0, Programmpaket zur Simulation des thermischen Verhaltens von Räumen, Raumgruppen und Gebäuden, © 1996 – 2007 K. Kreč (2007)
- [19] [THESIM V2.0, Programm zur mehrdimensionalen Thermischen Simulation eines Raums, © 1996 – 2007 K. Kreč \(2007\)](#)

See also: [Theoretical background](#)

Remarks to EN ISO 13786 (2000)

This essay is available in German only.

Anmerkungen zur EN ISO 13786 (2000)

In diesem Anhang werden Inhalte der Norm ÖNorm EN ISO 13786 [1] analysiert und diskutiert. Hier bei wird generell auf die derzeit gültige Ausgabe vom 1. 8. 2000 Bezug genommen. Eine Besonderheit dieser Ausgabe ist, dass die Formeln zur Definition und zur Berechnung der wirksamen Wärmespeicherkapazitäten in der österreichischen Fassung dieser Norm durch ein nationales Vorwort umformuliert werden. Auf die Untersuchung der Hintergründe und Notwendigkeiten für diese Maßnahme wird hier besonderes Gewicht gelegt...

Lesen Sie hier weiter:

- [Anmerkungen zur EN ISO 13786 \(pdf\)](#)



The above essay have been incorporated with friendly permission from *Klaus Kreč, Büro für Bauphysik, A-3562 Schönberg am Kamp, Veltlinerstr. 9, Österreich/Austria* .

[1] ÖNorm EN ISO 13786: Wärmetechnisches Verhalten von Bauteilen – Dynamisch-thermische Kenngrößen – Berechnungsverfahren, Ausgabe: 2000-08-01 (2000)

[2] ON V 31: Katalog für wärmeschutztechnische Rechenwerte von Baustoffen und Bauteilen, Ausgabe: 1. 12. 2001 (2001)

See also: [Theoretical background](#)

Calculating the specific heat capacity of an equivalent component

This essay is available in German only.

Berechnung der spezifischen Wärmekapazität der Ersatzkonstruktion

Der im Folgenden dargelegte Algorithmus erlaubt es, die spezifische Wärmekapazität der als Ersatzkonstruktion dienenden fiktiven, homogenen Schicht derart zu berechnen, dass die flächenbezogene wirksame Wärmekapazität dieser homogenen Schicht genau der flächenbezogenen wirksamen Wärmekapazität des mehrdimensional berechneten Ausschnitts der untersuchten Ziegelwand entspricht.

Die flächenbezogene wirksame Wärmekapazität: Als Definition für die wirksame Wärmekapazität wird der Ansatz des nationalen Vorworts der ÖNorm EN ISO 13786 [1] in der Form ...

Lesen Sie hier weiter:

- [Berechnung der spezifischen Wärmekapazität der Ersatzkonstruktion \(pdf\)](#)



The above essay have been incorporated with friendly permission from *Klaus Kreč, Büro für Bauphysik, A-3562 Schönberg am Kamp, Veltlinerstr. 9, Österreich/Austria* .

[1] ÖNorm EN ISO 13786: Wärmetechnisches Verhalten von Bauteilen – Dynamisch-thermische Kenngrößen – Berechnungsverfahren, Ausgabe: 2000-08-01 (2000)

[2] Bartsch, H.J.: Taschenbuch mathematischer Formeln, Verlag Harri Deutsch Thun, Frankfurt am Main (1982)

See also: [Theoretical background](#)

The effective thermal heat capacity

This essay is available in German only.

Die wirksame Wärmespeicherkapazität

Als zentraler Begriff für die Beschreibung der Wärmespeicherfähigkeit von Baukonstruktion-en wird die sog. „wirksame Wärmespeicherkapazität“ in verschiedenen Normenwerken verwendet. Aus diesem Grund ist es notwendig, sich mit dieser Kenngröße kritisch zu befassen und deren Aussagekraft zu analysieren ...

Lesen Sie hier weiter:

- [Die wirksame Wärmespeicherkapazität \(pdf\)](#)



The above essay have been incorporated with friendly permission from *Klaus Kreč, Büro für Bauphysik, A-3562 Schönberg am Kamp, Veltlinerstr. 9, Österreich/Austria* .

- [1] Kreč K.: Bewertung der Sommertauglichkeit von Gebäuden, Forschungsbericht im Auftrag der Initiative Ziegel im Fachverband der Stein- und keramischen Industrie, 31. März 2006
- [2] Stockinger, J., Kreč, K. & Seidl, J.: Thermisch optimierter Einsatz von Ziegel in Passivhäusern, Tagungsbericht der 8. Europäischen Passivhaustagung 2004, 205-219, Krems (2004)
- [3] Stockinger, J.: Thermisch optimierter Einsatz von Ziegel im Passivhaus, Master-Thesis, Donau-Universität Krems (2004)
- [4] Bruckmayer, F.: Der praktische Wärme- und Schallschutz im Hochbau, Deuticke-Verlag Wien (1949)
- [5] Koch H. A. & Pechinger, U.: Möglichkeiten zur Berücksichtigung von Sonnen- und Wärmestrahlungseinflüssen auf Gebäudeoberflächen. Gesundheits-Ingenieur 98, H. 10, S. 265-280, (1977)
- [6] [Heindl W., Kreč K., Panzhauser, E. & Sigmund, A.: Wärmebrücken. Springer-Verlag Wien-New York \(1987\)](#)
- [7] ÖNorm B8110-1, Wärmeschutz im Hochbau – Anforderungen an den Wärmeschutz und Nachweisverfahren. (2000)
- [8] EN ISO 10211-1: Wärmebrücken im Hochbau – Wärmeströme und Oberflächentemperaturen – Teil 1: Allgemeine Berechnungsverfahren. (1996)
- [9] [Kreč, K.: Zur Wärmespeicherung in Baukonstruktionen. Gesundheits-Ingenieur 114, H. 1, S. 11-18, \(1993\)](#)
- [10] ÖNorm EN ISO 13786: Wärmetechnisches Verhalten von Bauteilen – Dynamisch-thermische Kenngrößen – Berechnungsverfahren, Ausgabe: 2000-08-01 (2000)
- [11] [Kreč, K.: Zur dreidimensionalen Simulation des thermischen Verhaltens von Gebäuden“, Gesundheits-Ingenieur 121, Heft 6, 293-344 \(2000\)](#)
- [12] ÖNorm B8110-3, Wärmeschutz im Hochbau – Wärmespeicherung und Sonneneinflüsse, Ausgabe: 1999-12-01 (1999)
- [13] EN 832, Wärmetechnisches Verhalten von Gebäuden – Berechnung des Heizenergiebedarfs – Wohngebäude (1998)
- [14] ÖNorm EN ISO 13791, Wärmetechnisches Verhalten von Gebäuden - Sommerliche Raumtemperaturen bei Gebäuden ohne Anlagentechnik - Allgemeine Kriterien und Validierungsverfahren (2004)
- [15] ÖNorm EN ISO 6946, Bauteile – Wärmedurchlasswiderstand und Wärmedurchgangskoeffizient – Berechnungsverfahren (2005)
- [16] Minke, G.: Lehm- und Ökobuch, Stauf bei Freiburg (1999)
- [17] [AnTherm V3.51, Programmpaket zur Analyse des Thermischen Verhaltens von Bauteilen mit Wärmebrücken, © 2004 – 2007 T. Kornicki \(2007\)](#)
- [18] GEBA V7.0, Programmpaket zur Simulation des thermischen Verhaltens von Räumen, Raumgruppen und Gebäuden, © 1996 – 2007 K. Kreč (2007)
- [19] [THESIM V2.0, Programm zur mehrdimensionalen Thermischen Simulation eines Raums, © 1996 – 2007 K. Kreč \(2007\)](#)

See also: [Theoretical background](#)

Thermal conductivity and heat storage within planar components (slabs)

This essay is available in German only.

Wärmeleitung und Wärmespeicherung in plattenförmigen Bauteilen

Instationäre Wärmeleitungsprobleme, die natürlich auch die Berücksichtigung der Wärmespeicherung beinhalten, wurden in [1] in relativ allgemeiner Weise behandelt. Die dort geschilderten Methoden erlauben eine praktische Anwendung nur unter Einsatz entsprechender Computer-Programme, wenn man von besonders einfachen Fällen absieht. Diese einfachen Fälle sind jene, in denen eindimensionale Wärmeleitung vorliegt, also Wärmeleitung durch plattenförmige Bauteile bzw. durch Kugel- oder Zylinderschalen.

Da plattenförmige Bauteile im Hochbau die Regel darstellen, scheint es der Mühe wert, für diese ein einfacheres Rechenverfahren anzugeben und zu diskutieren. Für stationäre Wärmeleitung sind diese Verfahren seit langem bekannt, für instationäre Wärmeleitung auch schon seit einigen Jahrzehnten - siehe z. B. [2, 3, 4, 5]. Diese Verfahren sollen nun in den größeren Zusammenhang der in [1] dargelegten Methoden gestellt werden...

Lesen Sie hier weiter:

- [Wärmeleitung und Wärmespeicherung in plattenförmigen Bauteilen \(pdf\)](#)



The above essay have been incorporated with friendly permission from Klaus Kreč, Büro für Bauphysik, A-3562 Schönberg am Kamp, Veltlinerstr. 9, Österreich/Austria .

[1] [Kreč, K.: Zur Wärmespeicherung in Baukonstruktionen. Gesundheits-Ingenieur 114, Heft 1, S. 11-18 \(1993\)](#)

[2] Carslaw, H.S. & Jaeger, J. C.: Conduction of heat in solids. Second edition, Oxford University Press (1959)

[3] Heindl, W.: Neue Methoden zur Beurteilung des Wärmeschutzes im Hochbau. Die Ziegelindustrie, Hefte 4, 5, 6 (1967)

[4] Masuch, J.: Untersuchung des thermischen Verhaltens klimatisierter Räume bei variabler innerer Temperatur. Dissertation; Berlin (1971)

[5] Masuch, J.: Analytische Untersuchungen zum regeldynamischen Temperaturverhalten von Räumen. VDI-Forschungsbericht 557 (1973)

[6] ÖNorm EN ISO 13786, Wärmetechnisches Verhalten von Bauteilen – Dynamisch-thermische Kenngrößen – Berechnungsverfahren, Ausgabe 2000-08-01 (2000)

[7] Eichler, F.: Praktische Wärmelehre im Hochbau. 4. Auflage, VEB-Verlag für Bauwesen, Berlin (1964)

[8] Berber, J.: Bauphysik – Wärmetransport-Feuchtigkeit-Schall, Verlag Handwerk und Technik (1973)

[9] Tschegg, E., Heindl, W. & Sigmund, A.: Grundzüge der Bauphysik, Springer-Verlag Wien New York (1984)

See also: [Theoretical background](#)

On the definition of the effective thermal heat capacity

This essay is available in German only.

Zur Definition der wirksamen Wärmespeicherkapazität

In diesem Anhang wird jener Ansatz für die wirksame Wärmespeicherkapazität hergeleitet, der im Vorwort der ÖNorm EN ISO 13786 [1] festgehalten ist...

Lesen Sie hier weiter:

- [Zur Definition der wirksamen Wärmespeicherkapazität \(pdf\)](#)



The above essay have been incorporated with friendly permission from *Klaus Kreč, Büro für Bauphysik, A-3562 Schönberg am Kamp, Veltlinerstr. 9, Österreich/Austria* .

[1] ÖNorm EN ISO 13786: Wärmetechnisches Verhalten von Bauteilen – Dynamisch-thermische Kenngrößen – Berechnungsverfahren, Ausgabe: 2000-08-01 (2000)

[2] ÖNorm B8110-3, Wärmeschutz im Hochbau – Wärmespeicherung und Sonneneinflüsse, Ausgabe: 1999-12-01 (1999)

See also: [Theoretical background](#)

Tutorials and Cookbooks

Tutorials shall help you to learn the AnTherm application quickly and efficiently.

Examples and guides collected below shall explain the usage of the application. They shall allow the user to quickly learn the operation of the application and contain useful tips regarding input strategies also.

Following tutorials are available:

- [Modelling and Evaluation Tutorial](#)
- [VAPOUR - Tutorial](#)
- [Usage Hints Introductory Tutorial](#)
- [Transiente Simulation - Tutorial](#)
- [Example images](#)
- [Example: Results report \(edited\)](#)
- [Example: Coupling Coefficients report \(edited\)](#)

Hint: The application help window will always display on top (in front of) all other application windows when invoked directly from the application (via F1 key or the Help menu) and eventually overlap them. To have help window be freely switchable via desktop manager (which will allow you to bring the application windows to front) you shall start the application help from the windows start menu instead (Start-All Programs-AnTherm-Help) and browse the contents manually to the chapter of your choice.

With the [application setting "Help Window shown TopMost"](#) the user can decide if the Help window will be shown topmost to all application windows and automatically minimize with the application (which is the windows default behaviour) or it can be set behind the application window via typical desktop window manager (default application setting).

Example project files

There are several example [projects](#) included within the application's installation to give you a quick start. You will find these examples in the *subfolder "Beispiele"* (means "examples" in german) of the *installation folder* (typically its is *C:\Program Files\tkornicki\AnTherm\Beispiele*). After opening one of these example projects (via menu [Help->Example Load...](#)) please save a working copy of it to your private folder (menu [File->Save as...](#)).

Below the listing of some:

BEISPIEL_1	3-dimensional modelling of an building corner with balcony. This example is identical to the 3 rd validation example of EN ISO 10211 .
BEISPIEL_2	3-dimensional modelling detail of a glass roof over indoor swimming pool. To avoid condensation one of the joists is heated by filament (heating power can be entered).
BEISPIEL_3	3-dimensional modelling detail of exterior wall of vertically perforated bricks. The calculation of heat transport is executed to obtain medium U-Value for unrendered wall.
DEMO1_3D	Wall corner made of homogenous material, plastered on both sides with concrete ceiling with plaster. Three-dimensional calculation.
DEMO3_3D	Masonry wall corner plastered on both sides with concrete ceiling with plaster. Three-dimensional calculation.
Hallenbad_2D	VAPOUR - Tutorial : Example of two dimensional heat and vapour transport (diffusion) and interstitial condensation risk assessment. Junction detail of flat roof of an indoor pool. Dividing wall between living room and indoor swimming pool: double skin brick wall with 5 cm core insulation. Calculation of the heat and vapour transport.
Hallenbad_3D	VAPOUR - Tutorial : Example of three dimensional heat and vapour transport (diffusion) and interstitial condensation risk assessment. Junction details of the flat roof over an indoor pool. Dividing wall between living room and indoor swimming pool: double skin brick wall with 5 cm core insulation. Exterior wall of the swimming pool: 25 cm concrete, 15 cm core insulation, 12.5 cm brick sheathing. Calculation of the heat and vapour transport
overlap	Example showing effect of overlapping principles of data entry. Flipping the order of elements results in different overlapping product. Try reordering (promote/demote) of a selected element.
TUTOR2_orig	TUTORIAL - 2-dimensional : Reinforced concrete slab with masonry bearing wall on girder over carport (+ exterior insulation and electric heating cable assembly embedded in concrete floor topping).
TUTOR3_orig	TUTORIAL - 3-dimensional : Corner construction over carport. Masonry bearing wall on girder over carport and bonded with concrete floor slab (+ ext. insul, perimeter insul., elect. heating assembly in floor topping).

EN ISO 10077	All conformance validation examples of EN ISO 10077 (Frames).
EN ISO 10211	All conformance validation examples of EN ISO 10211 (Thermal bridges).
EN ISO 13786	All validation examples of EN ISO 13786 (Dynamic thermal indicators).

Remark: To quickly reach examples installed within the example folder use the [Menu Help->Example Load...](#).

Remark: Further tutorials, references, examples, videos and imagery of AnTherm can be available at the AnTherm's website or Blog also. Please visit them frequently. You will find the link to AnTherm's web within the [Help menu](#).

See also: [Introduction](#), [Data Entry & Input windows](#), [Evaluations and Results](#)

Modelling and Evaluation Tutorial

The program operations shall be illustrated here with step-by-step instructions for analysing a demonstration object. The procedures of input and evaluation can be followed as described to generate and evaluate two- and three-dimensional input models equivalent to the examples contained in the distribution. These completely calculated model files, which can be perused before working with this program directly, also serve as an object of comparison and aid towards successful completion of the model as described in the following.

Note: To quickly reach examples contained in the distribution you shall use [menu Help->Example Load...](#).

> Start reading with "[Object of Analysis](#)"...

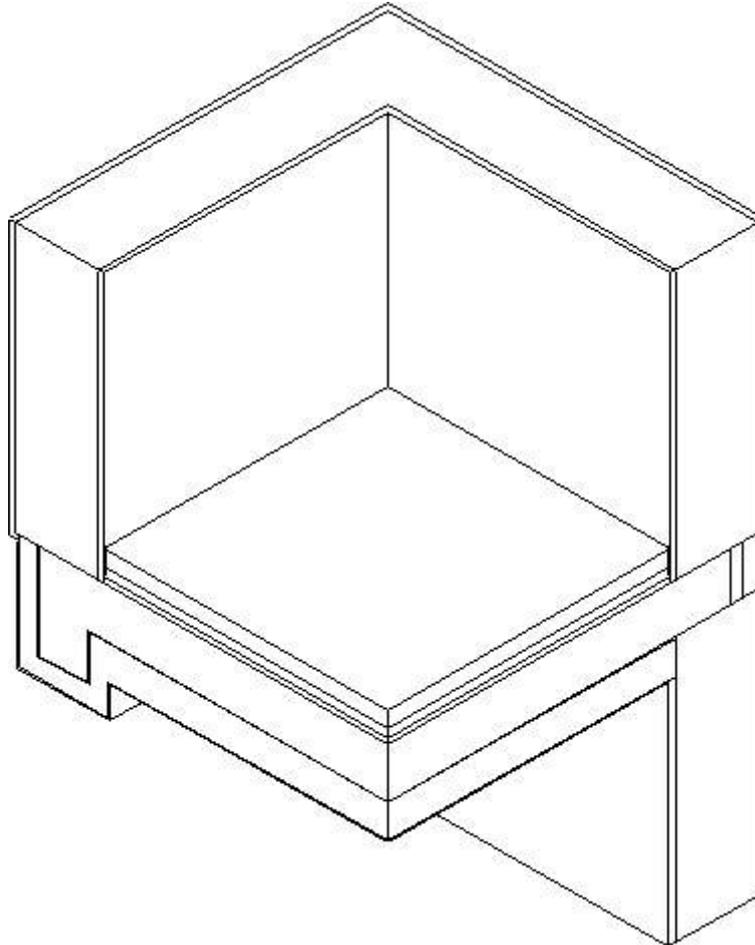
Hint: The application help window will always display on top (in front of) all other application windows when invoked directly from the application (via F1 key or the Help menu) and eventually overlap them. To have help window be freely switchable via desktop manager (which will allow you to bring the application windows to front) you shall start the application help from the windows start menu instead (Start-All Programs-AnTherm-Help) and browse the contents manually to the chapter of your choice.

With the [application setting "Help Window shown TopMost"](#) the user can decide if the Help window will be shown topmost to all application windows and automatically minimize with the application (which is the windows default behaviour) or it can be set behind the application window via typical desktop window manager (default application setting).

See also: [Usage Hints Introductory Tutorial](#)

Object of Analysis

The demonstration object to be modelled and evaluated encompasses the corner region of a masonry and concrete construction over an exterior space (e.g. carport). As illustrated in the axiometric (axonometric) rendering below, the masonry bearing wall is bonded with the concrete floor slab along the side which flanks the exterior space underneath, and supported by a girder spanning an opening in the other direction.



The standard masonry wall construction is a total of 35.5 cm thick, composed of (from the exterior):

- 2.5 cm stucco
- 30.0 cm masonry
- 2.0 cm interior plaster

In the region flanking the floor slab, the masonry wall is substituted by:

- 2.5 cm stucco
- 6.0 cm perimeter brick
- 4.0 cm perimeter insulation
(floor slab)

The reinforced concrete girder supporting the other side of masonry is encased in 7.0 cm of standard insulation.

The floor construction over the exterior space is built up to a total thickness of 41.5 cm with:

- 0.5 cm exterior plaster
- 12.0 cm insulation
- 0.5 cm adhesive mortar

- 17.5 cm reinforced concrete
 - 2.0 cm sand cushion
 - 3.0 cm isolating underlayment*
 - 1.0 cm embedded heating assembly (concrete)*
 - 5.0 cm concrete topping*
- *- expansion strip along outer perimeter (0.4 cm)

Despite extensive exterior insulation, such a prominent corner situation could prove to be a critical thermal bridge. A potential heat source has also been included to simulate an electrical floor heating assembly in the concrete topping. The efficiency of such a heating alternative in this exposed location can thus be assessed as well.

> Continue reading with "[Stages of Analysis](#)" ...

Stages of Analysis

The most critical cross-section of the object under consideration is through the wall/floor connection at the girder. This section shall be modelled and evaluated first in the two-dimensional mode of the program. The results obtained here would be valid for the region of the thermal bridge in which heat flow can be expected to occur strictly parallel to the section plane, i.e. at least approximately 1 m from the corner. Instructions for this application are provided in the next chapter to this tutorial, Two-dimensional Analysis.

In order to obtain a picture of the temperature distribution in the apparent thermal bridge at the outer corner of the construction, a three-dimensional model of the situation must be generated for evaluation. The procedures of this stage are treated in the chapter on [Three-dimensional Analysis](#).

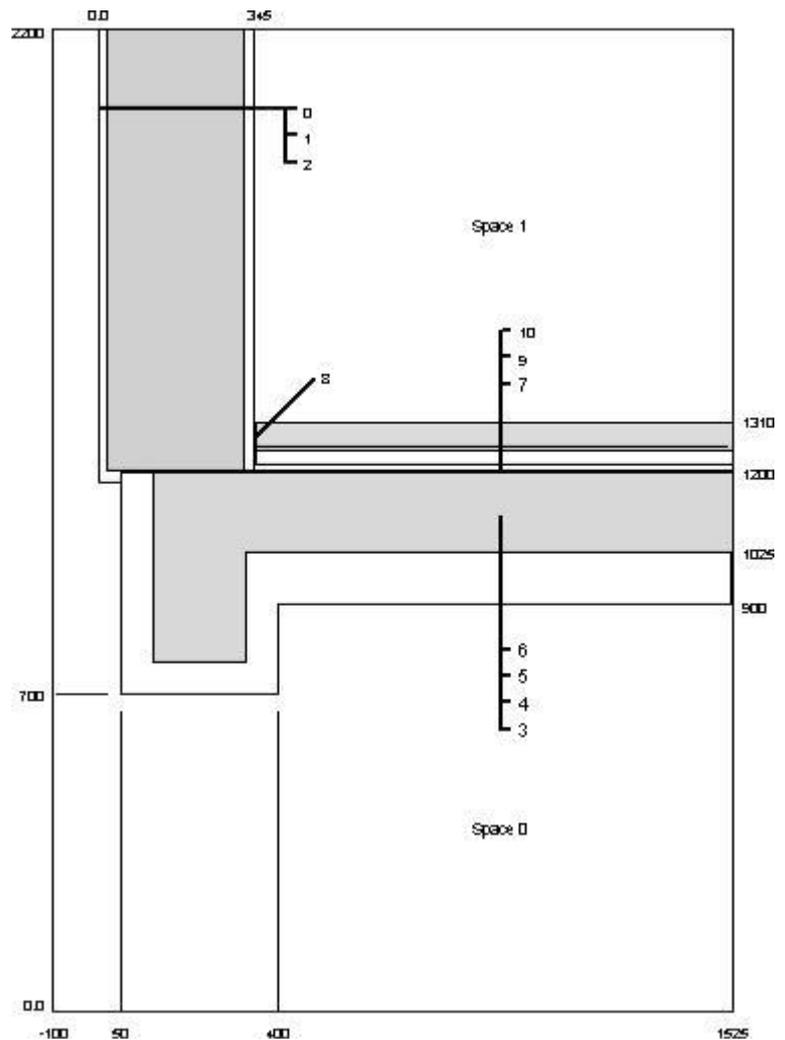
To facilitate input in three-dimensional mode, the previously generated two-dimensional model can serve as the first layer (component group) and reference basis for further component group input. The transformation of an object from two to three dimensions by means of the input branch of the program is outlined in the section on [Generating the Model - Input / Transforming an Existing 2-Dimensional Model](#).

> Continue reading with "[Two-dimensional Analysis](#)" ...

Two-dimensional Analysis Modelling the Object

Before actually working with the program, a sketch of the object should be prepared in which relevant data is summarised for model input. Such a sketch of the two-dimensional demonstration object would include the information shown below: dimensions, reference coordinates, and material properties.

	material	λ [$\text{Wm}^{-1}\text{K}^{-1}$]
0	stucco	0.800
1	masonry	0.164
2	interior plaster	0.700
3	exterior plaster	0.800
4	insulation	0.041
5	adhesive mortar	0.271
6	reinf. concrete	2.200
7	sand	1.000
8	expansion strip	0.150
9	isol. underlayment	0.040
10	concrete topping	1.200
	space	α [$\text{Wm}^{-2}\text{K}^{-1}$]
0	exterior	25
1	interior	6



Since negative coordinate values are allowed, the positioning of the graph origin ($x=0$, $y=0$) is generally arbitrary. The reference coordinates shown here, however, are the ones consistent with the geometric input data as related in the next section.

Continue reading with "[Generating the Model - Input](#)" ...

Generating the Model - Input

The program is called up with the Start-Menu *Start→Programs→AnTherm→AnTherm*, whereupon the default screen configuration appears.

From the main menu select *File→New→2D Project*. In the Description window enter some text information identifying the current project, e.g.

```
TUTORIAL - 2-dimensional:
Reinforced concrete slab with masonry bearing wall on girder over
carport (+ exterior insulation and electric heating cable assembly
embedded in concrete floor topping).
```

Note: Whatever text is typed here will be included at the head of program printouts (model documentation and numeric results). This screen can be accessed from the main window at any time by selecting *View→Windows→Description*.

Make sure Element input windows are open. There shall be at least Elements2D and ElementsBrowser widows already open now. If not, go into *View-Menu* and select *View→Primary Edit Windows*.

You can close the *Elements 3D* and *Layer Browser* window if displayed, by clicking on X at its top-right corner, since those windows are not being referenced in this tutorial.

The first element to be entered is the exterior space. On the *ElementsBrowser* window click *New button*. The element Edit window (a Box Editor) shall reveal and a new, empty element shall list in the elements' list.

The boundaries of element 1 shall demarcate the full extent of the model with

```
X_1: -100 X_2: 1525
Y_1: 0 Y_2: 2200
```

- Start entering the coordinates by entering -100 for x1 first.
- Then press TAB-key to move the input focus to x2.
- Enter 1525 for x2 and TAB to next field, which is y1.
- Enter 0 for y1 and TAB to y2.
- Enter 2200 for y2 and confirm again with TAB.

Having now the focus in the "Box Type" field press arrow-down key to select "Space Box" as the type of this element (or either use the mouse to select the requested type).

Now TAB twice (skipping the Group entry for now) to move the input focus into "Surface Name" field. Enter "exterior surface" and TAB to its alpha value. Enter 25 and confirm the entry with the TAB again.

Name the room with "exterior" which will be later used to setup the temperature for this boundary. Confirm with TAB.

Once these entries are confirmed, element 1 is complete and the program is ready for input of the next element.

Now it would be worthwhile to re-scale the control graph for a better view of the component during input of successive elements: press *Fit* on the *Element2D window* and let the application redraw the graph.

Note to earlier WAEBRU users: As opposite to Waebru, AnTherm's Element 2D View does always show all elements of the one (and within the 2D model the only) layer. There is no such option to select the object display level to aggregate. Naming of groups and grouping is done directly to each element.

Before setting out to build up the material component, some thought should be given to the sequence of element input so as to best utilise the advantages of element overlapping. To illustrate this principle, the demonstration example described here has been organised into construction parts of predominantly parallel material layers. This simplifies coordinate input substantially.

The first part of the model to be entered is the wall structure. An "onion skin" sequence begins at the outermost layer (stucco) with the second element:

```
X_1: 0
X_2: 1525
Y_1: 1175
Y_2: 2200
```

On the *ElementsBrowser* window click *New button* again. The element Edit window (the Box Editor) shall reveal and a new, empty element shall list in the elements' list next to the one entered before.

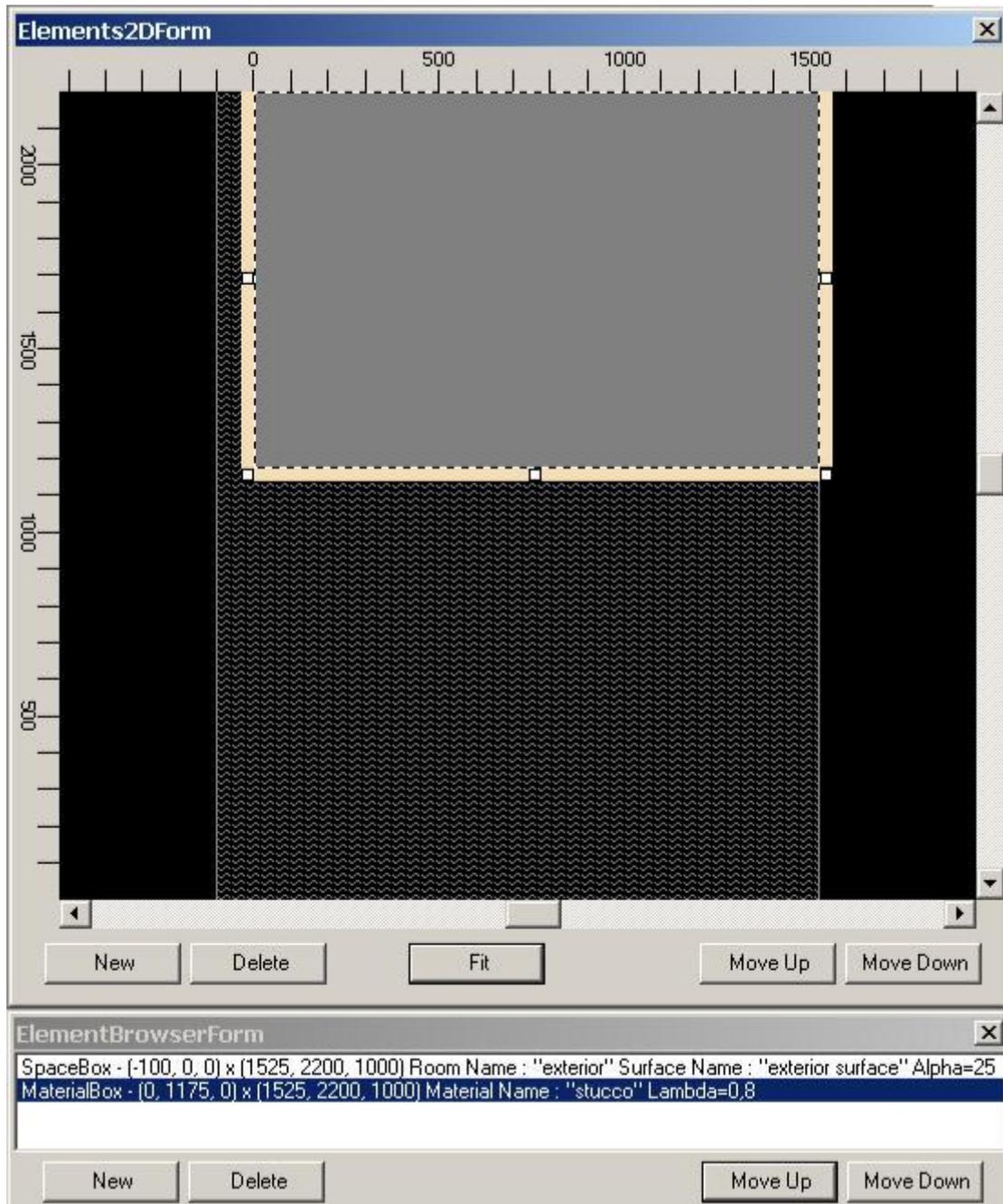
Enter the coordinates and confirm each entry with a TAB key as before. Using arrow keys while the focus is at the Box Type select "Material Box".

TAB to *Material Name* and enter *stucco*.

TAB to *Lambda* field and enter *0.8*.

Note: While using international features windows provides you with country specific setting for decimal comma. This shall allow you to use numeric pad of your keyboard for entering number correctly. Please observe, that entering 0,8 (with comma instead of dot) might result in 08 if your setting of current operating system locale (typically Regional Setting of the System's Control Panel) defines different character (for example a dot) for decimal comma. The opposite might also be valid (e.g. german system use comma - the , character) for decimal comma. If the system is set up properly for your country, the numeric pad's comma will provide the right character to the input.

The control graph should now look like this:



To insert next new element use the *context menu* available to the *Box Editor*. Over the *Box Editor* click the *right mouse button* to reveal the *Edit context menu*. Select *Add→Append*.

The next element is the masonry wall, which overlaps - and thereby displaces - all but a thin strip of stucco (25 mm to the left and bottom):

```
X_1: 25
X_2: 1525
Y_1: 1200
Y_2: 2200
Box type: Material Box
Material name: Masonry wall
Lambda: 0.164
```

TAB to x1 and proceed with entering the data as before.

The interior plaster, the fourth element, delineates the right side of the masonry.

Instead of using *New* on *ElementsBrowser* or the *context menu's Add→Append* sequence to add that next element to the list use "Duplicate" on the *Box Editor window*. Since you have just TABbed to it and it is visible marked as focused, simply press ENTER to execute the Duplicate operation.

Here, only one coordinate of the set still visible from element 3 needs to be adjusted to leave 200 mm of masonry thickness:

```
X_1: 325
X_2: 1525
Y_1: 1200
Y_2: 2200
Box type: Material Box
Material name: Interior plaster
Lambda: 0.7
```

TAB to x1 and proceed with entering the data as before.

When you confirm the entry and TAB to "Duplicate" press ENTER receiving again a copy of the lately active (selected) element. This simplifies the input to just the need of changing the things which are different for the new element and also leaves your hand off the mouse.

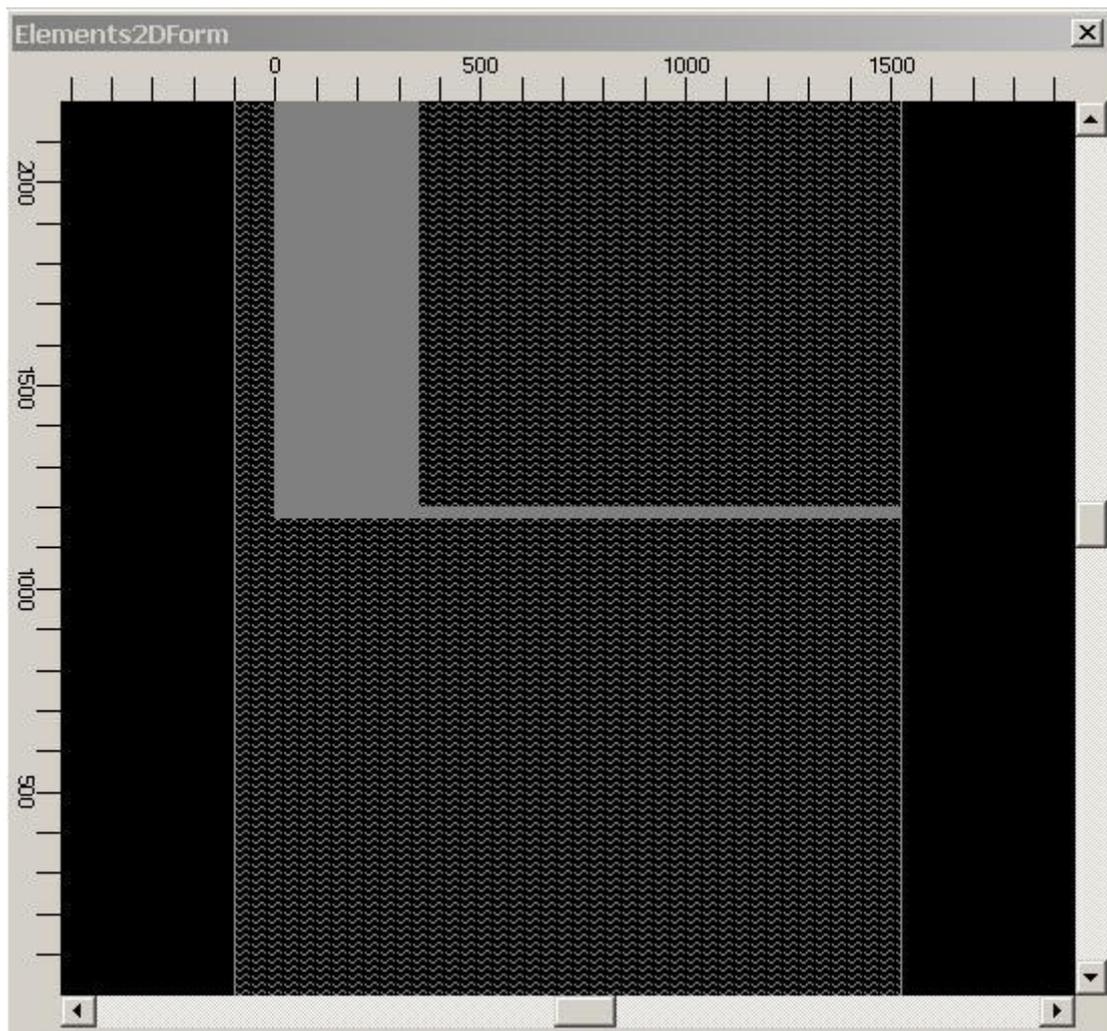
The next new element is the "kernel" of this part of the aggregate: the interior space overlapping all but the total wall thickness (with extended strip of stucco) of the material entered thus far:

```
X_1: 345
X_2: 1525
Y_1: 1200
Y_2: 2200
Box type: Space Box
Surface name: interior surface
Alpha: 6
Room name: interior
```

Create new element, TAB to x1 (if needed) and proceed with entering the data as before.

The last element edited is shown selected in the *ElementsBrowser* window. Holding the *control-key click* over the line with that selected element to deselect it. The selection mark shall disappear.

The control graph should now display the following figure of the standard masonry construction:



Note: The illustration above shows only the silhouette of the component in its current state.

The second part of the construction to be entered is the insulated girder, followed by the concrete floor slab with its exterior insulation. The final part of input covers the interior floor construction over the slab. These sequences are covered in the section on **Completing the Component** further on in this tutorial.

Continue reading with ["Completing the Tables"...](#)

Completing the Tables

With the over-all sequence of input in mind, the table of building materials can be completed in one uninterrupted input series. Once complete, the list of available material properties can be accessed from *Materials window* to facilitate material identification and input.

Any of the elements of type Material Box can then be subsequently assigned using the material table.

Select *View*→*Windows*→*Material Window* to reveal the list of Materials. If the list is empty we will have to re-enter materials used so far:

Right-click on the Material window and select *Add Material* from the context menu shown. An template of a material is added to the list. *Right-click* at that new line in the list - the Material Editor shall appear

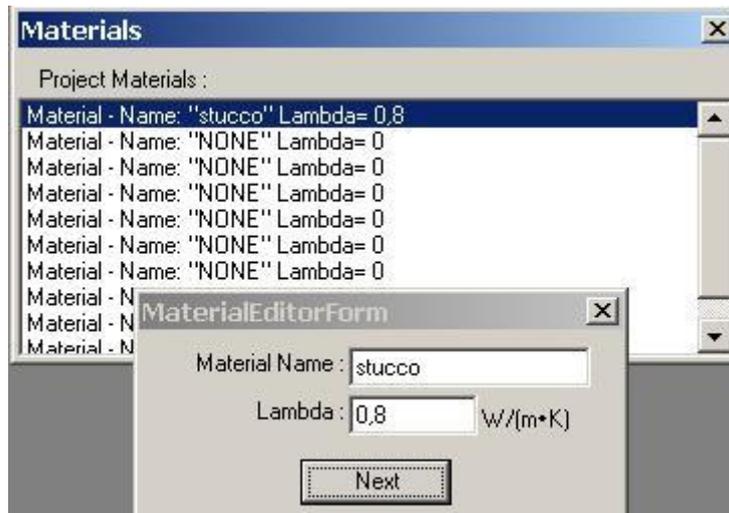
Click at Material Editor and TAB to *Material Name* field.

Enter "stucco" and TAB to Lambda field.

Enter "0.8" and TAB to confirm the entry.

Right-click on the Material window and select *Add Material* from the context menu shown. You can add as many templates as you expect to need. In this tutorial we will use 11 different materials, thus add further templates (*repeat Right-click on the Material window* and select *Add Material*) to the list.

Finally, having now further 10 material templates, move the list to make "stucco" visible (using scroll bar if required) and *right-click on it* to get *Material Editor* show it again.



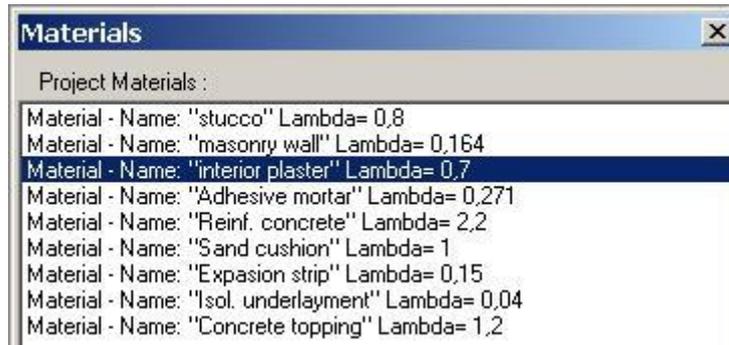
Click "Next" on Material Editor. The first template is selected and shown.

TAB to Material Name and enter "masonry wall". TAB to Lambda and enter "0.164". TAB to "Next" confirming the entry and *press ENTER* (which does the same as clicking Next).

On the third line now selected, TAB to Material Name and continue with entry of:

Material name	Lambda
Interior plaster	0,7
exterior plaster	0.8
insulation	0.041
Adhesive mortar	0.271
Reinf. concrete	2.2
Sand cushion	1.0
Expansion strip	0.15
Isol. underlayment	0.040
Concrete topping	1.2

To have a printout of the complete tables at hand as a reference during input, use Data-Entry Report from the View→Window menu.



The data associated with a particular material can be edited by selecting it in the Material list. Individual entries can be deleted from the table screen directly (*mark line and right-click*, then selecting *Delete Material* from the context menu).

Note: Material list just simply provide a container holding materials for reference. Material properties of elements are currently stored by elements themselves, thus either changes apply only to the related object (element) and does not influence the other (or another element).

Note: The method described above for completing the materials table can also be used to generate the other type of tables - surfaces - for later selection.

Continue reading with "[Completing the Component](#)"...

Completing the Component

To move on to input of the remaining elements, click *New* on *Elements Browser* or *Box Editor* window. The screen should then appear ready for input of element 6, now empty or a copy of the last element.

The next sequence of four overlapping elements describes the girder part of the aggregate:

elem. 6, ext. plaster:

```
X_1: 45
X_2: 405
Y_1: 695
Y_2: 1200
elem. type :Material
material: exterior plaster
```

After having x1,x2,x3,x4 entered and selecting the type of "Material Box", instead of entering material data, simply *double click* the "interior plaster" in the "Materials" window. The material data is being copied into the fields.

By *double-clicking a material* in *Materials window* while a material box is selected, you assign the material data to the currently selected material element.

Note: you can also *drag a material from Materials window onto the Box Editor* to achieve the same result.

Note: If the *Materials window* is hidden or not shown you can *reveal it by double-clicking the label "Material Name" or "Lambda" or the accompanying data entry fields* (shown when element type is "Material Box").

Continue with the data entry and assignment of material properties as follows:

elem. 7, ext. insulation:

```
X_1: 50
X_2: 400
Y_1: 700
Y_2: 1200
elem. type :Material
material: insulation
```

elem. 8, adh. mortar:

```
X_1: 120
X_2: 330
Y_1: 770
Y_2: 1200
elem. type :Material
material: adhesive mortar
```

elem. 9, reinf. concrete:

```
X_1: 125
X_2: 325
Y_1: 775
Y_2: 1200
elem. type :Material
material: reinf. concrete
```

Elements 10 through 13 describe the same sequence of materials for the insulated concrete floor slab construction connected to the girder:

elem. 10, ext. plaster:

```
X_1: 400
X_2: 1525
Y_1: 895
Y_2: 1200
elem. type :Material
material: exterior plaster
```

elem. 11, ext. insulation:

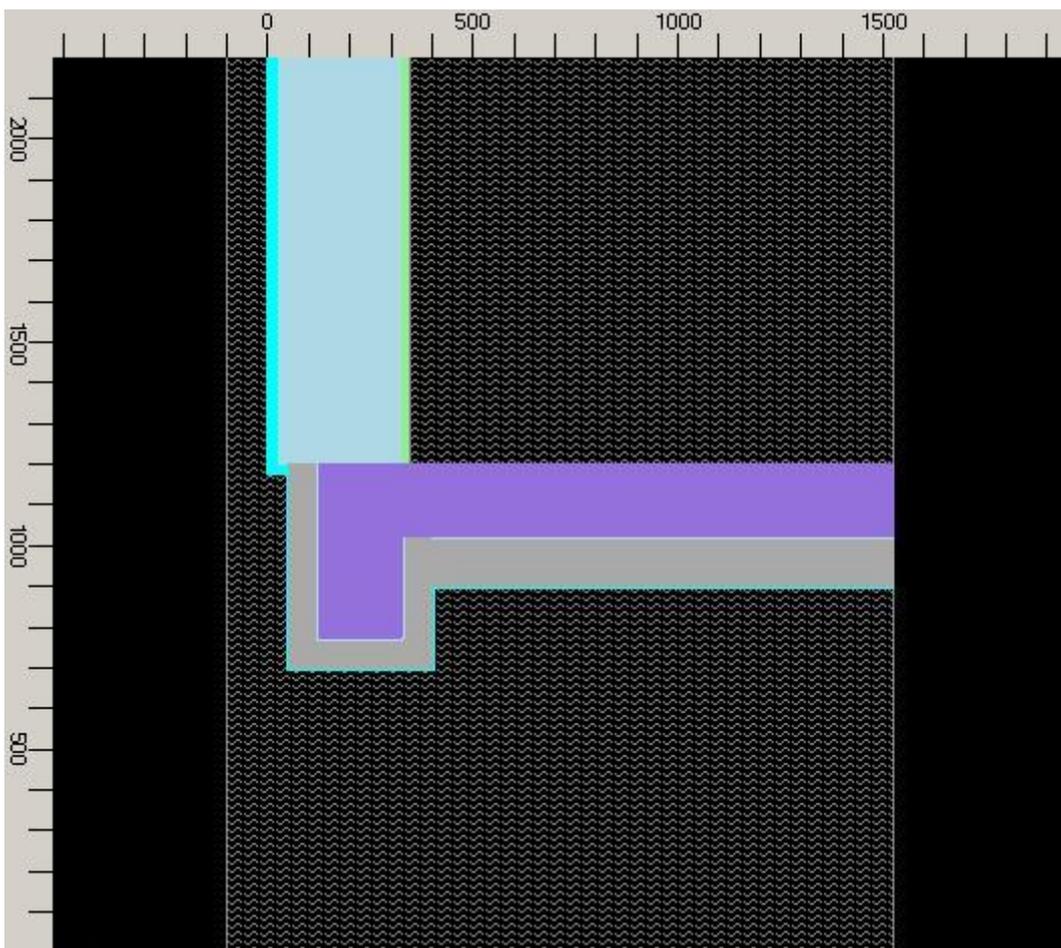
```
X_1: 400
X_2: 1525
Y_1: 900
Y_2: 1200
elem. type :Material
material: insulation
```

elem. 12, adh. mortar:

```
X_1: 400
X_2: 1525
Y_1: 1020
Y_2: 1200
elem. type :Material
material: adhesive mortar
```

elem. 13, reinf. concrete:

```
X_1: 325
X_2: 1525
Y_1: 1025
Y_2: 1200
elem. type :Material
material: reinf. concrete
```



The interior flooring over the concrete slab is built up with the last five elements of the aggregate:

elem. 14, sand:

```
X_1: 345
X_2: 1525
Y_1: 1200
Y_2: 1310
elem. type :Material
material: sand
```

elem. 15, exp. strip:

```
X_1: 345
X_2: 1525
Y_1: 1220
Y_2: 1310
elem. type :Material
material: exp. strip
```

elem. 16, isol. layer:

```
X_1: 349
X_2: 1525
Y_1: 1220
Y_2: 1310
elem. type :Material
material: isol. layer
```

elem. 17, concrete (then wrapped with heat/power source):

```
X_1: 349
X_2: 1525
Y_1: 1250
Y_2: 1310
elem. type :Material
material: concrete topping
```

The concrete shall also be attached to the power source (heating source). This is being done by wrapping the assigned area by the Power Box type of an element - as described in the reference, a Power Box has no interior, it does wrap interiors of Material Boxes.

To get exactly the same rectangle assigned to the next element as that of element 17, select *Duplicate from the context edit menu* (by *right-clicking* either at the last *element in Element Browser* or *right-clicking over Box Editor* and selecting *Duplicate*)

elem. 18, concrete wrapper with heat/power source:

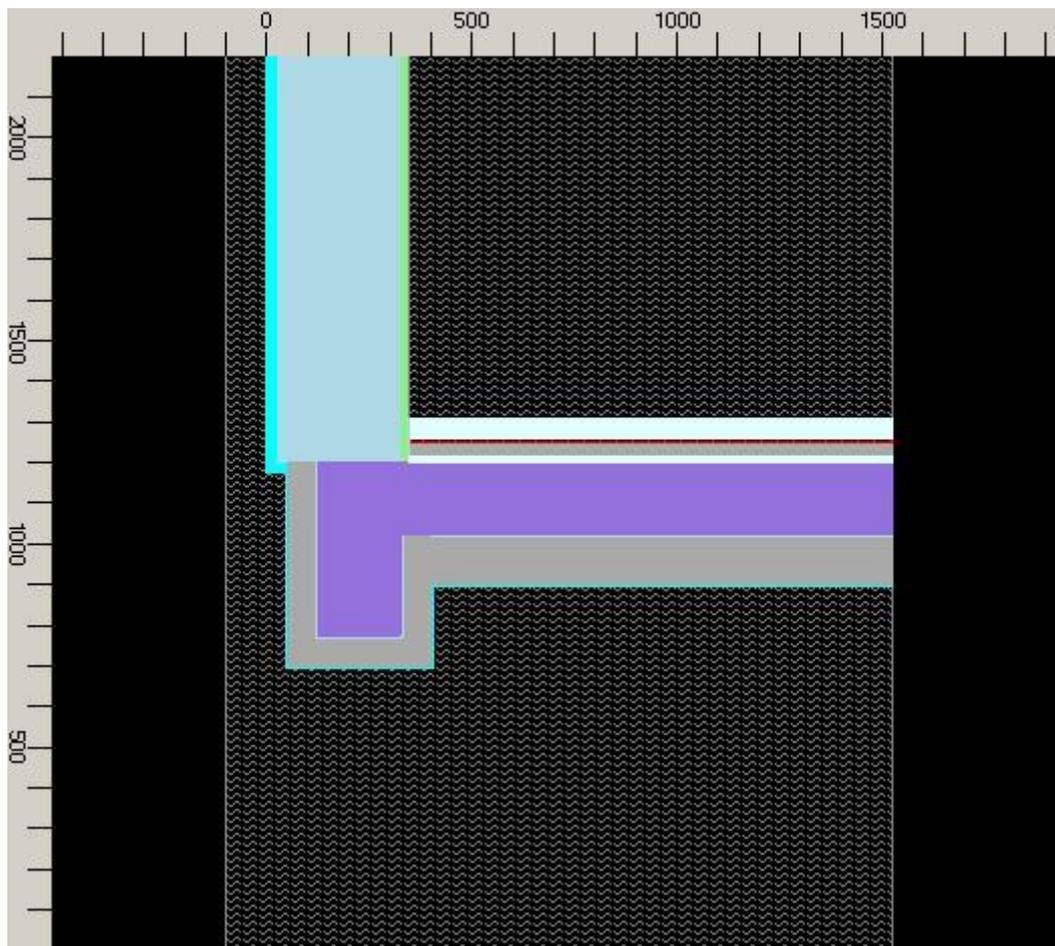
```
X_1: 349
X_2: 1525
Y_1: 1250
Y_2: 1310
elem. type :Power Box
power name: concrete
```

TAB to *Box Type* and change the entry to "Power Box". TAB to *Power(source) name* and enter "concrete" thus naming the *Power Source Boundary Condition* which can later be assigned with Power.

After confirming the entry of this Power Source continue with the last material element:

elem. 19, concrete topping:

```
X_1: 349
X_2: 1525
Y_1: 1260
Y_2: 1310
elem. type :Material
material: concrete topping
```



This concludes the entry of the two-dimensional model.

To verify the input, select *View* → *Windows* → *Data-Entry Report* and check details of the input.

Data-Entry Report

Demonstration License - exclusive demonstration preview
 Any production, duplication or disclosure is prohibited
 Copyright (c) Kormicki, Dienstleistungen in EDV und IT
 http://antherm.eu/ email: antherm@kormicki.com

AnTherm V.10.137.2
 (c) Kormicki, all rights reserved

21.02.2022

TUTORIAL - 2 dimensional:
 Reinforced concrete slab with masonry bearing wall on girder over carport (+ exterior insulation and electric heating cable assembly embedded in concrete floor topping).

File: T:\Unordnung\Entw2\Benchmark\TUTOR2_round.antherm

Details of the construction's input data

L.No.	No.	C	X1	Y1	Z1	X2	Y2	Z2	DX	DY	DZ	T	Details
1	1		-100	0	0	1525	2200	1000	1625	2200	1000	S	Floor/wall
1	2		0	1175	0	1525	2200	1000	1525	1025	1000	M	Stucco
1	3		25	1200	0	1525	2200	1000	1500	1000	1000	M	Masonry wall
1	4		325	1200	0	1525	2200	1000	1200	1000	1000	M	Int. plaster
1	5		345	1200	0	1525	2200	1000	1180	1000	1000	S	Room 1 / Int. transfer coeff.
1	6		45	695	0	405	1200	1000	360	505	1000	M	Exterior plaster
1	7		50	700	0	400	1200	1000	350	500	1000	M	Insulation
1	8		120	770	0	330	1200	1000	210	430	1000	M	Adhesive mortar
1	9		125	775	0	325	1200	1000	200	425	1000	M	Reinf. concrete
1	10		400	895	0	1525	1200	1000	1125	305	1000	M	Exterior plaster
1	11		400	900	0	1525	1200	1000	1125	300	1000	M	Insulation
1	12		490	1020	0	1525	1200	1000	1125	180	1000	M	Adhesive mortar
1	13		325	1025	0	1525	1200	1000	1200	175	1000	M	Reinf. concrete
1	14		345	1200	0	1525	1310	1000	1180	110	1000	M	Sand cushion
1	15		345	1220	0	1525	1310	1000	1180	90	1000	M	Expansion strip
1	16		349	1220	0	1525	1310	1000	1176	90	1000	M	Isol. underlayment
1	17		349	1250	0	1525	1310	1000	1176	60	1000	M	Concrete topping
1	18		349	1250	0	1525	1310	1000	1176	60	1000	P	PS 0
1	19		349	1260	0	1525	1310	1000	1176	50	1000	M	Concrete topping

Seite 1 von 1

72%

Obtaining basic results - evaluation

First characteristics of the component can be obtained immediately without any special calculations - materials and properties used within the model plus characteristic U-Values at profiles near to adiabatic boundaries.

To receive this basic documentation select *View*→*Data Input&Entry*→[Modelling Report](#) and check characteristics of the model:

The screenshot shows the 'Modelling Report' window in AnTherm. The report includes contact information for Kornicki, Dienstleistungen in EDV und IT, and the date 21.02.2022. It describes a 2-dimensional tutorial model of a reinforced concrete slab with masonry bearing wall. The report is divided into two sections for layered constructs and U-Value calculations.

Room 0 <-> Room 1 @ TopBack: (0, 2200, 0) x (345, 2200, 0)

Material / Surface	λ [W/(m·K)]	d [mm]	R_s [m ² K/W]	α [W/m ² K]	R [m ² K/W]
Room 0/Ext. transfer coeff.			0,0400	25,0000	0,0400
Stucco	0,8000	25,0000			0,0313
Masonry wall	0,1640	300,0000			1,8293
Int. plaster	0,7000	20,0000			0,0286
Room 1/Int. transfer coeff.			0,1667	6,0000	0,1667
	Σ	345,0000			U-Value: 0,4772 [W/m ² K]

Room 0 <-> Room 1 @ BackRight: (1525, 895, 0) x (1525, 1310, 0)

Material / Surface	λ [W/(m·K)]	d [mm]	R_s [m ² K/W]	α [W/m ² K]	R [m ² K/W]
Room 0/Ext. transfer coeff.			0,0400	25,0000	0,0400
Exterior plaster	0,8000	5,0000			0,0063
Insulation	0,0410	120,0000			2,9268
Adhesive mortar	0,2710	5,0000			0,0185
Reinf. concrete	2,2000	175,0000			0,0795
Sand cushion	1,0000	20,0000			0,0200
Isol. underlayment	0,0400	30,0000			0,7500
PS 0/Concrete topping	1,2000	10,0000			0,0083
Concrete topping	1,2000	50,0000			0,0417
Room 1/Int. transfer coeff.			0,1667	6,0000	0,1667
	Σ	415,0000			U-Value: 0,2464 [W/m ² K]

To receive further results some calculation is required.

Activate the *main menu Results...* option.

You will be asked to save the project data and to provide fine-grid and solver parameters. Confirm all the three dialogs with Ok. Default parameters for fine-grid and solver are sufficient for most cases and thus to complete this tutorial.

The construction file of a model must be accompanied by a grid file for calculation in the next branch of the program. Based on standard parameters, a grid file of the current model is generated. The calculation branch is automatically started. Convergence behaviour of the iterative calculation process is displayed in the form of running status messages.

As soon as the calculation of base solutions has been terminated successfully, the [Conductance-Matrix-Report](#) is generated and shown. Also a [Boundary-Conditions window](#) is revealed to receive your input for further evaluation steps - covered just below.

Number of evaluated cells: 1489

Thermal conductance [W / K]			
Room\Room	exterior	interior	
exterior		0,919387	
interior	0,919287		

Distribution factors [I]			
Powers.\Room	exterior	interior	
concrete	0,082475	0,917509	

Precision information			
	Close-up error [W / K]	Conduct. Sum [W / K]	Cundactance rel. close-up error
exterior	-1,00566e-004	0,919287	-1,09395e-004
interior	1,00566e-004	0,919387	1,09383e-004

	Close-up error [I]
concrete	1,57986e-005

Note: The conductance matrix results shown here are only "borderline acceptable" in terms of precision. For some purposes, a re-calculation of the base solutions with a more stringent termination condition may be called for at this point, before continuing with evaluation. For a demonstration object, however, the precision of these results will suffice.

The table of heat source distribution factors is printed out separately. These results imply that the heat source, i.e. the simulated electric heating assembly meant to heat space "concrete", would lose approximately 8% of its power to the exterior space - without flooring over the concrete topping.

Note: Just as the conductance matrix, the heat source distribution factors are condition-independent, i.e. valid for any temperatures and source powers of evaluation.

Activate *View*→*Windows*→[Psi-Value Determination](#) menu.

Psi-Value Determination

Space to assume exterior: Room 0
 Another space to assume interior: Room 1

U-Value Profiles at adiabatic cut-off planes
 Room 0 <-> Room 1 @ BackRig Room 0 <-> Room 1 @ TopBac

Layered construct Layered construct
 U-Value: 0.24644 W/m²K U-Value: 0.47715 W/m²K
 Length: 1525 mm Length: 1305 mm
 U * l: 0.37582100 W/mK + U * l: 0.62268075 W/mK = Total U * l: 0.99850175 W/mK

Determine Lengths by Reference Point
 Determine Reference Point
 at exterior dimensions (Room 0)
 at interior dimensions (Room 1)
 by manual input

Leitwert 2D : 0.922472 W/mK
 Room 0
 Room 1
Psi Value : 0.076030 W/mK

X: 0 mm
 Y: 895 mm

Calculating the Psi-Value (linear thermal transmittance) characteristic for that component is still independent from any boundary conditions too.

Obtaining results dependant on particular boundary conditions (space air temperatures and power source intensities) requires the input of that data at first:

BoundaryConditionsForm

Name	Value	Unit
▶ exterior	-10	°C
interior	20	°C
concrete	0	W/m ²

Apply

The heat source is thus "turned off" for the first round of evaluation. Confirm the set of boundary conditions by clicking at *Apply* on the [Boundary Conditions window](#).

After a short waiting period, during which the temperature distribution in the component is calculated by the program, the following results are shown in [Results-Report](#).

Number of evaluated cells: 1489 (Nodes = 18423)

Boundary conditions and resulting Surface Temperatures / Condensing Humidity

	Air temperature [°C]	min.temperature [°C]	max.temperature [°C]	Condensing.H. [%]	f_{Rsi}^*
exterior	-10,00	-9,97	-9,38	100,00 %	
interior	20,00	15,24	18,49	74,06 %	0,84
	Power [W]	Power density [W/m ²]	Volume [m ³]		
concrete	0,00	0,00	0,01176000		

Weighting factors for coldest surface point of each room

	exterior	interior
g(concrete)	0,000128	0,065438
g(exterior)	0,999131	0,158732
g(interior)	0,000869	0,841268

Coordinates (x,y,z) for coldest surface point of each room

	x	y	z	Temp.[°C]	f_{Rsi}^*
exterior	45,0000	695,0000		-9,97	
interior	345,0000	1310,0000		15,24	0,84

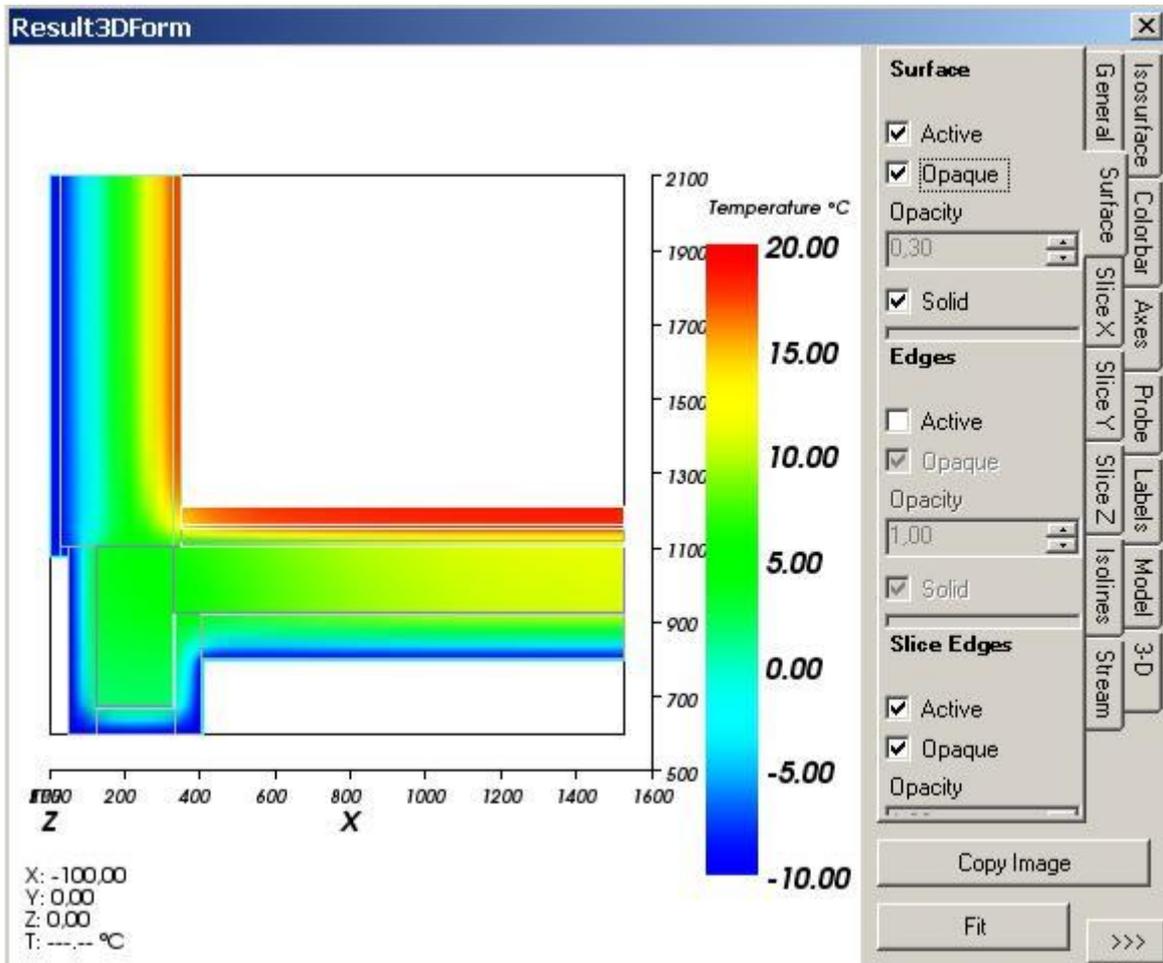
Note: Though weighting factors (g-Values) as a whole are not dependant on boundary conditions, they are included here in association with the coldest surface points (the location of which is in general condition-dependant).

> Continue reading with "[Graphic results](#)" ...

Graphic results

Graphic results will be obtained in the sophisticated Results-3D-View which is commonly used for two- and three-dimensional output. If yet not visible go to to view menu and select Results 3D View to reveal the graphics evaluation window.

Cause we currently analyse a two-dimensional case, make sure perspective view is turned off in the general panel shown. Also you can turn full opacity of the surface with the surface panel.



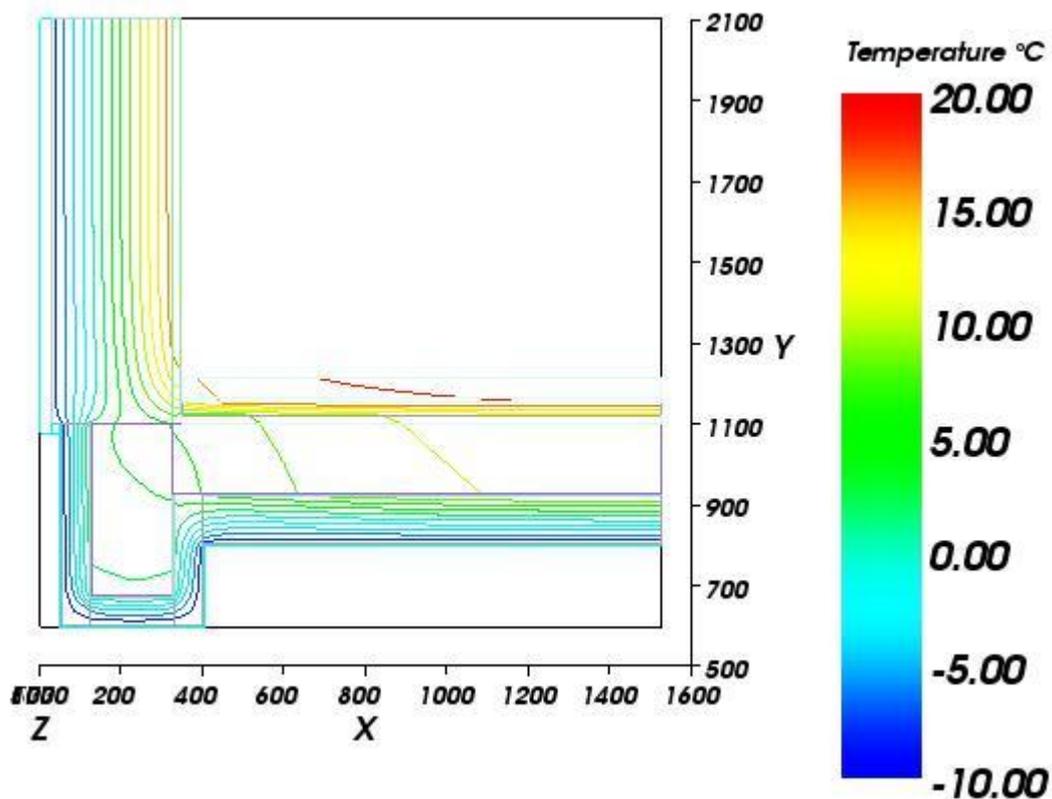
To pass the image for farther processing, press Copy Image. The image is being placed into the clipboard from where you can insert it into any application for report generation or high quality printing. You can adjust the size of the image produced within application settings of the edit menu.

Isolines (Isotherms)

Next, a rendering of the temperature distribution in the form of isotherms shall be generated using the panel Isolines (for Isotherms).

Cause Isolines plotted will be plotted over the surface already visible thus rendering them in the same color would make them invisible, let the surface be fully transparent by deselecting "Opaque" and changing the opacity to 0 on the surface panel.

After selecting "Active" on the Isolines panel, enter 2.0 for the step (interval) specification.



To pass the image for further processing via clipboard use either Copy Image or Edit Menu→Copy.

Annotated (Labelled) Isolines (Isotherms)

Next, a rendering of the temperature distribution in the form of annotated (labelled) isotherms shall be generated using the Z Slice.

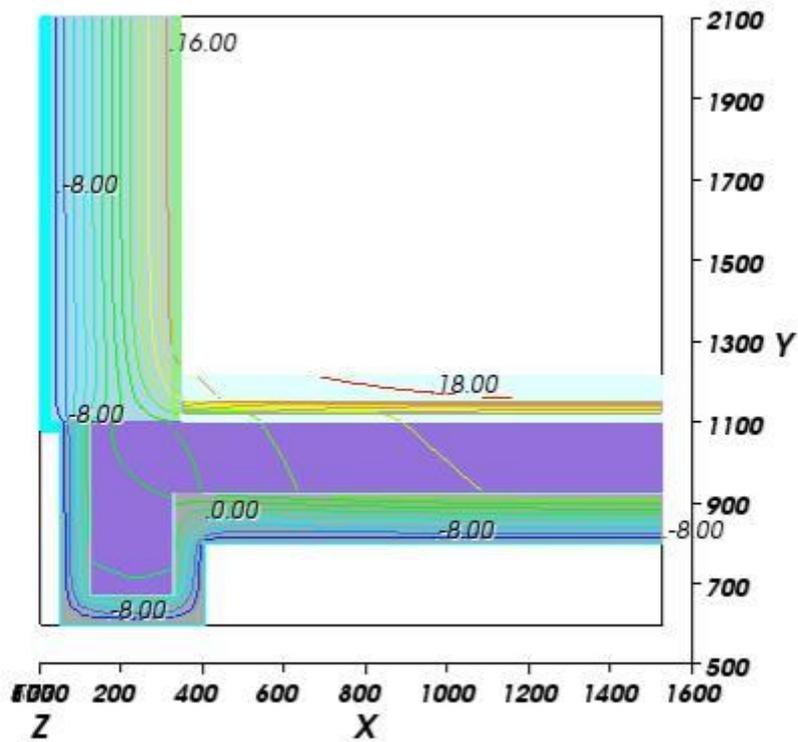
Since the Slice placed "within" the construction it would render invisible if the surface would not be turned off. Therefore on the surface panel make the surface inactive by turning it off.

On Slice Z panel select the value of 500.

The slice surface is also coloured. To reveal isotherms only, make the slice fully transparent by turning its opacity value to 0.00.

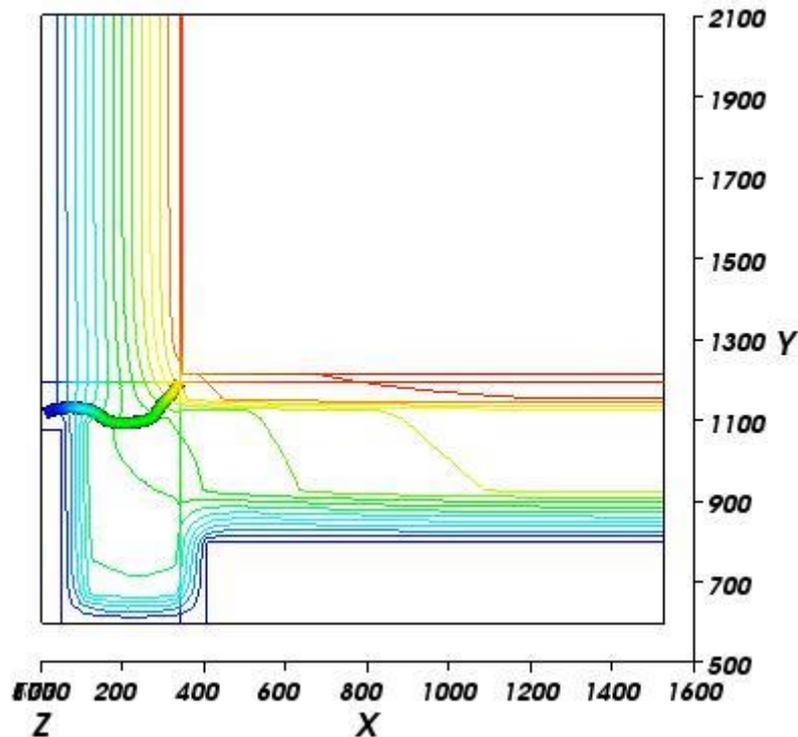
On the labels panel adjust the label distance and size parameters.

To reveal the structure of materials within the model, now shown as lines only, select Model Slices to be Solid on the Model Panel.



Streamline analysis

Before going to the streamline analysis, turn Model completely off (both Model and Model Slices). Using the Slice X and Slice Y controls place the crossing of both somewhere within the construction revealing the streamline. On the Streamline panel make the tube radius larger to emphasize the streamline shown.

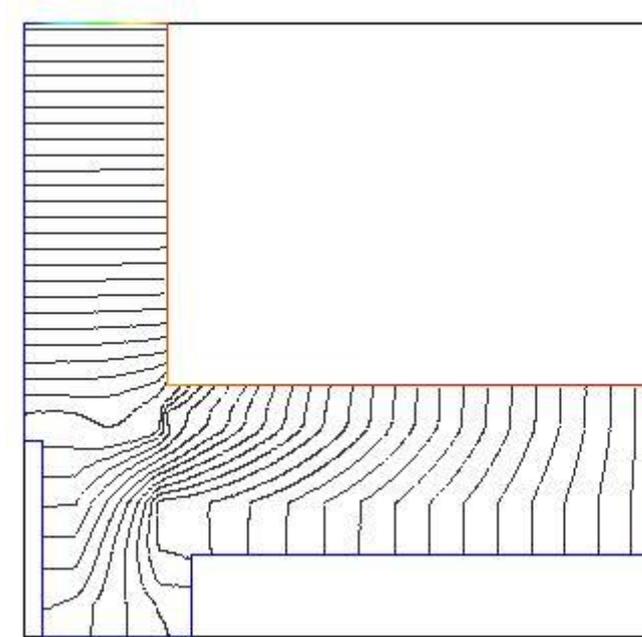


You can optionally turn Isolines or Labels off for less obscured representation.

Streamline analysis - multiple streamlines (2D only)

There is also an option to show multiple streamlines at the Z-Slice of this 2-dimensional model. To achieve this select "Start at: Space Boundary" option on the Streamline panel. By default the heat flow of the first room is subdivided into 5 intervals. At boundaries of this intervals streamlines are drawn.

The diagram below was drawn with 50 intervals for heat flowing into exterior space. Changing the tube radius of streamlines to 0 renders them as thin lines. Turn colorize of streamlines to black by clicking on colorize check box once (second click would render the white) We've also turned isolines and axes views, the slice X and Y off.



Note: Since this model contains only two spaces, the heat flowing out of interior space is the same as the heat entering exterior space. Selecting another space would therefore result in same lines being drawn.

This chapter is not finished yet. More evaluation examples shall be included here later on.

> Continue reading with "[Comparing Results under Different Conditions](#)"....

Comparing Results under Different Conditions

The same model shall now be evaluated under the condition of an active heat source element. The electric heating assembly embedded in the floor is assumed to have a power output of 60 W/m^2 . In the two-dimensional simulation, which is modelled as a layer of 1 m thickness, this output corresponds to 60 Watts per meter length of heating element in the section. The power density of such 10 mm thick heating assembly is then:

$$60 \text{ W/m}^2 / 0.010 \text{ m} \rightarrow 6000 \text{ W/m}^3$$

and this is what has to be provided as input to the program. This also show how to calculate the power density based on manufacturer data and construction entered.

Alternatively one could calculate the total power of the source and divide it by the resulting volume. The total power to be assigned to the heat source here is therefore:

$$60 \text{ W/m}^2 \times 1.176 \text{ m} \times 1.000 \text{ m} \rightarrow 70.56 \text{ W}$$

Having that and the total volume of the power source (this can be obtained [from earlier basic reports](#)). On the other hand, for the two-dimensional simulation being modelled as a layer of 1 m thickness the power density of the whole heat source results in:

$$70.56 \text{ W} / 0.01176 \text{ m}^3 \rightarrow 6000 \text{ W/m}^3 .$$

or

$$70.56 \text{ W} / (1.176 \text{ m} \times 1.000 \text{ m} \times 0.010 \text{ m}) \rightarrow 70.5 \text{ W} / 0.01176 \text{ m}^3 = 6000 \text{ W/m}^3 .$$

and this is what hat to be provided as input to the program and same as the value computed earlier above.

To re-define the conditions of evaluation, activate the window *Boundary conditions* (if hidden turn to *View*→*Evaluation*→*Boundary Conditions*). Move the cursor to the power input field (the temperature values can be left as is) and enter the new power density value [W/m^3]:

BoundaryConditionsForm			
	Name	Value	Unit
	exterior	-10	°C
	interior	20	°C
▶	concrete	6000	W/m ³

Confirm the entry with *Apply*.

Numeric Results

Since the conductance matrix and heat source distribution factors remain the same for any given evaluation conditions, these need not be printed out again.

The Evaluation results printout, however, does show new data:

Number of evaluated cells: 1489 (Nodes = 18423)

Boundary conditions and resulting Surface Temperatures / Condensing Humidity

	Air temperature [°C]	min.temperature [°C]	max.temperature [°C]	Condensing.H. [%]	f_{Rsi}^*
exterior	-10,00	-9,96	-9,16	100,00 %	
interior	20,00	17,61	27,81	86,15 %	0,92
	Power [W]	Power density [W/m ²]	Volume [m ³]		
concrete	70,56	6000,00	0,01176000		

Weighting factors for coldest surface point of each room

	exterior	interior
g(concrete)	0,000128	0,000014
g(exterior)	0,999131	0,079543
g(interior)	0,000869	0,920457

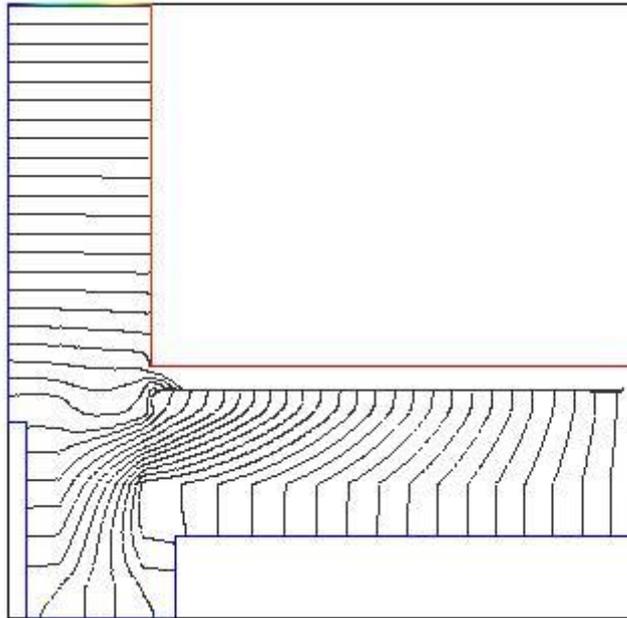
Coordinates (x,y,z) for coldest surface point of each room

	x	y	z	Temp.[°C]	f_{Rsi}^*
exterior	45,0000	695,0000		-9,96	
interior	345,0000	2166,2500		17,61	0,92

The coldest interior surface point is no longer at the floor/wall corner. The temperature on the floor surface is also considerably warmer than without the heat source (nearly 8° higher).

Graphic Results

Having the graphic results window open from the previous evaluation the significant change could be observed.



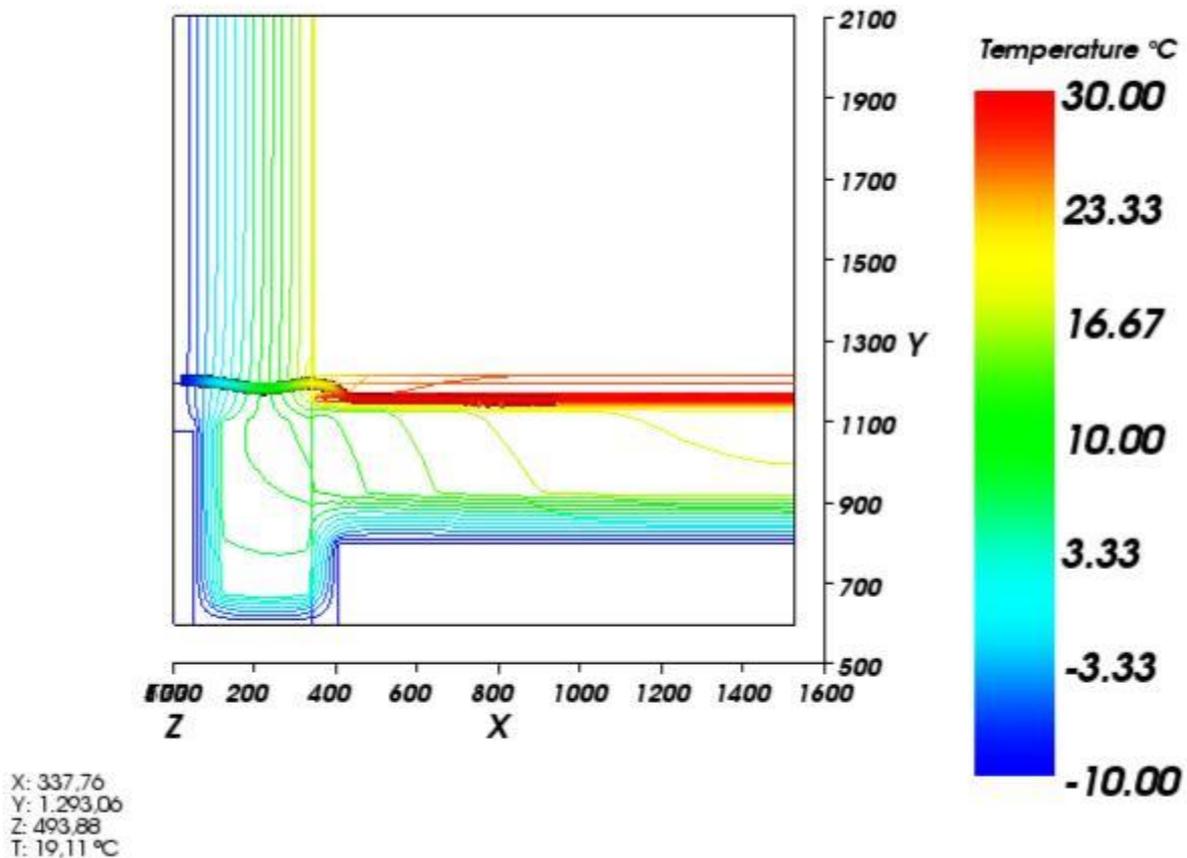
Note different shape of streamlines drawn!

Let's change back to single streamline view (turn all Slices on, select streamline to start at Slice X/Y/Z and eventually change the tube radius back to earlier value of about 12 and colorizing on).

Now turn the isolines back on to see the temperature distribution.

This time the evaluation range of temperatures is noticeably broader. As could be expected, the isotherm lines, drawn at the same temperature interval as before, are highly concentrated in the insulation.

Please go to Isotherms and adjust the evaluation range properly to reflect the change. Also the Colorbar needs some adjustment for this new, extended, range of temperatures to be coloured sufficiently. This is a matter of choice if you want the same colour structure to be kept for comparative analysis with the earlier (powerless) evaluation or not.



This chapter is not finished yet. More evaluation examples shall be included here later on.

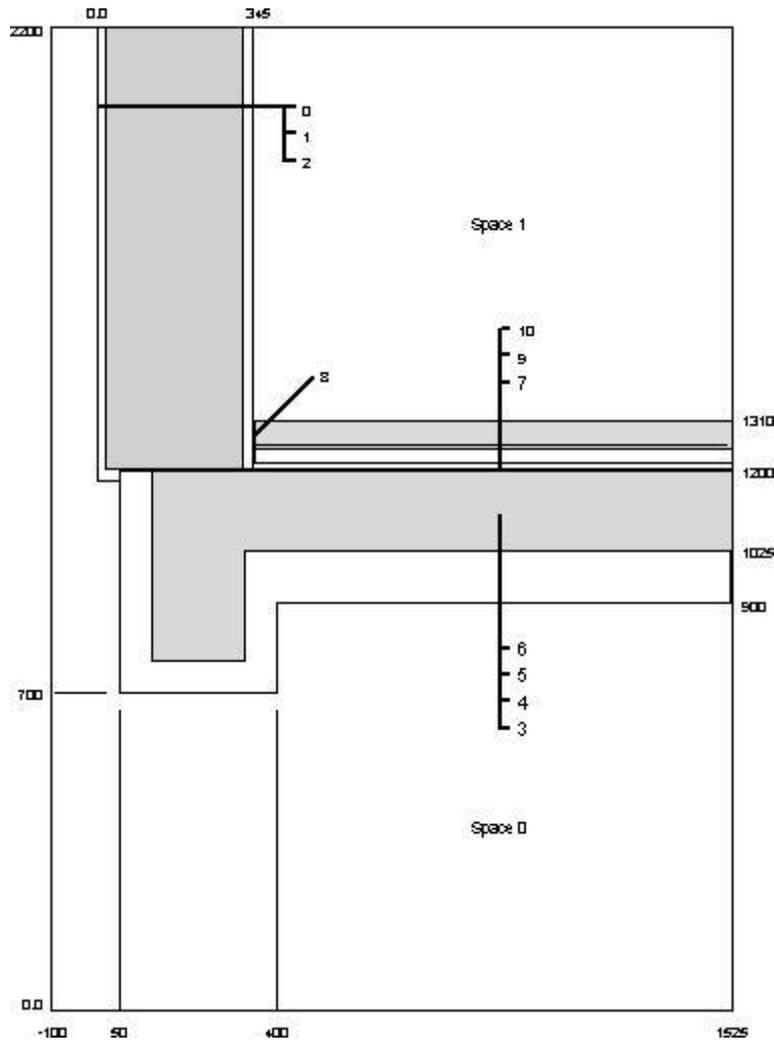
> Continue reading with "[Three-dimensional Analysis](#)"...

Three-dimensional Analysis Modelling the Object

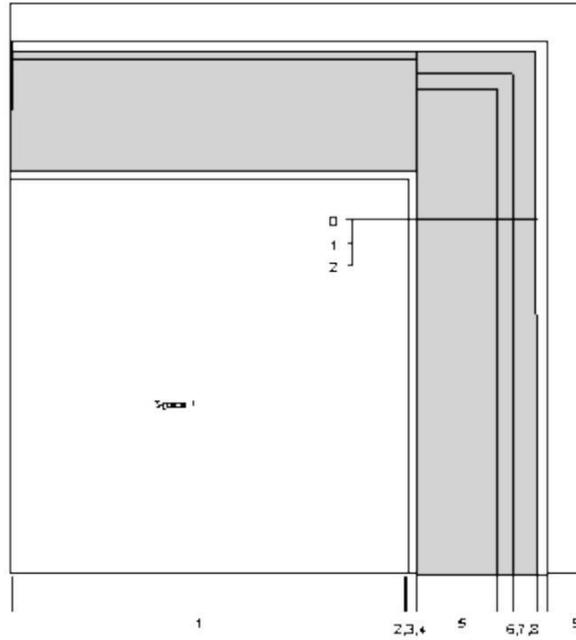
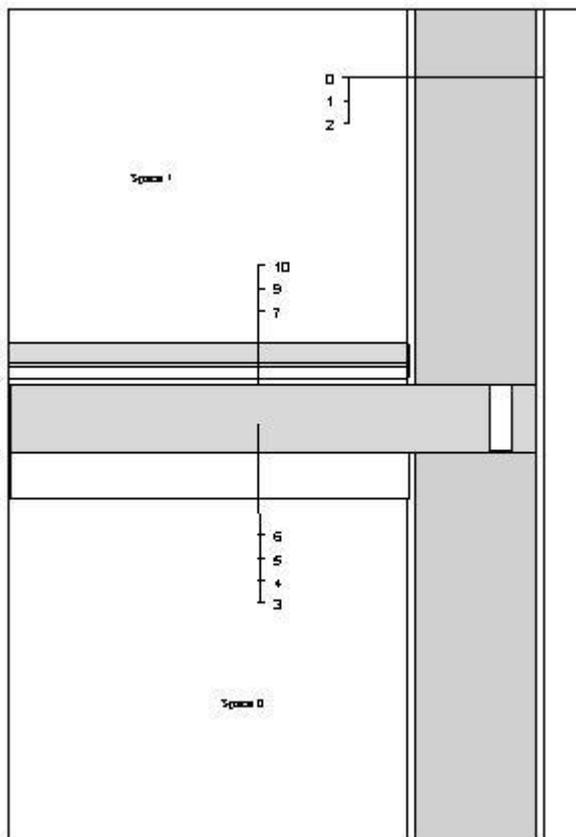
Three-dimensional objects are entered as a series of component groups, each describing a two-dimensional section of the object for a specified thickness. A sketch of the model should therefore be made as an orientation aid for input of layers in the z direction.

Such an orientation sketch should show the object in orthogonal sections, including reference coordinates consistent in all three directions. For the three-dimensional demonstration object, the sketch would show the geometry as follows:

x/y section



see [sketch of 2-dimensional model!](#)

x/z section**z/y section**

The reference list of material properties for the two-dimensional section modelled in the last chapter covers nearly all of the materials required here. The only new material is that of the perimeter brick, for which $\lambda = 0.500$.

layers	thickness
1	1175
2	1
3	4
4	20
5	200
6	40
7	60
8	25
9	100

> Continue reading with "[Generating the Model - Input](#)"...

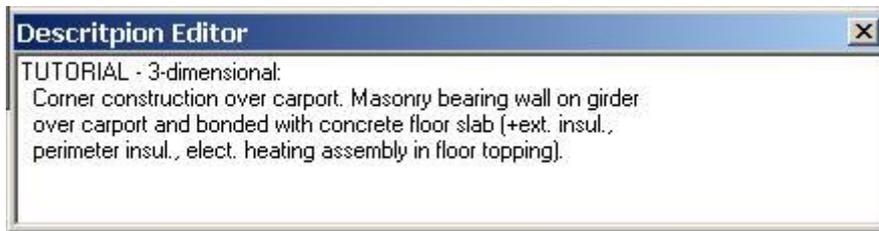
Generating the Model - Input

As already mentioned, the three-dimensional input model of the demonstration object need not be generated "from scratch". A section of the first layer in the z direction already exists in the form of the [two-dimensional model](#), including nearly [complete tables](#). These files can be transformed and used as the basis for the model to be generated in this chapter.

Transforming an Existing 2-Dimensional Model

In order to load two-dimensional files for editing in three-dimensional mode, the input branch of the program has a *Convert menu*. Assuming the 2D model generated in the last chapter was named *tutor2*, load it into the program via *File→Open* and then choose *File→Convert→Layer 3D Project*.

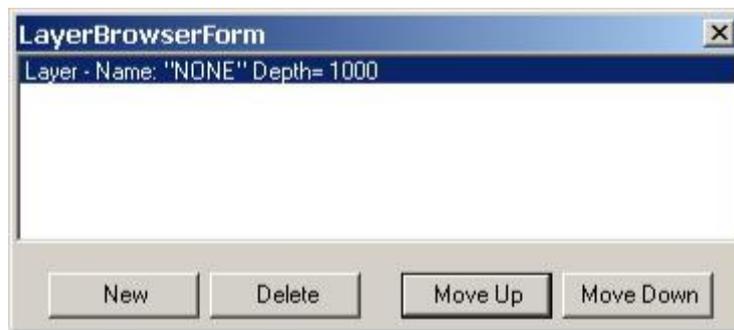
The project description shows with the text as it was entered for the two-dimensional files. This can be appropriately edited here, e.g.



Input can be continued from here as described below. Before leaving the input branch, however, the model must be explicitly saved in 3D files which are accessible for the rest of the program package in three-dimensional mode.

To save the intermediate data into the new project file select *File→Save As...* and apply a new name (e.g. *tutorial3*).

Within the Layer Browser double-click onto the (currently) only layer



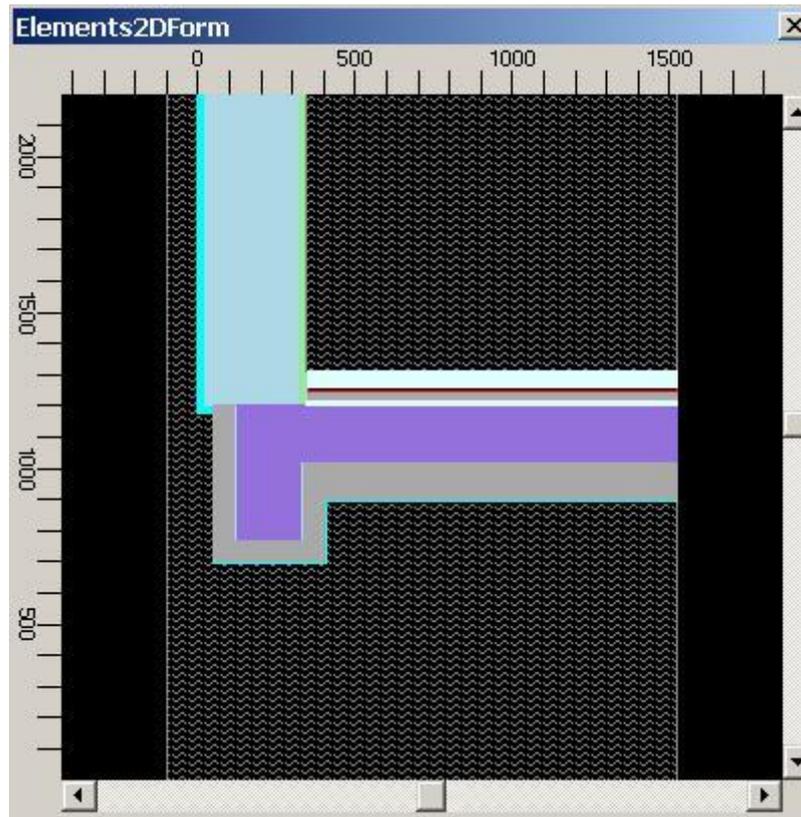
to open the Layer Editor and enter layer data:

Layer Name :Floor/wall Depth.:1175



Use TAB-key to move between the input fields and to confirm the data entered.

Element 2D view still shall show the same picture as of the two-dimensional model since we have not made any changes to the layer 1 except of its thickness which is perpendicular to the view shown.



To secure the data select *File*→*Save* which immediately will write to the file named *tutotr3* as defined and the beginning of this section.

Select *File*→*Exit* to terminate the program now.

The formerly two-dimensional model is now transformed and ready for completion. To run the entire program package for continued work on this model either start the program from the *Windows Start Menu* or double click onto the file *tutor3.antherm* which shall start the program with the data already read in.

> Continue reading with "[Completing the Model in Three Dimensions](#)"...

Completing the Model in Three Dimensions

The construction will be built up by copying and editing component layers in succession.

Starting with layer 1, floor/wall section, displayed as illustrated above (use view menu), *right-click over the Layer Browser* to reveal its context menu and select *Duplicate*.

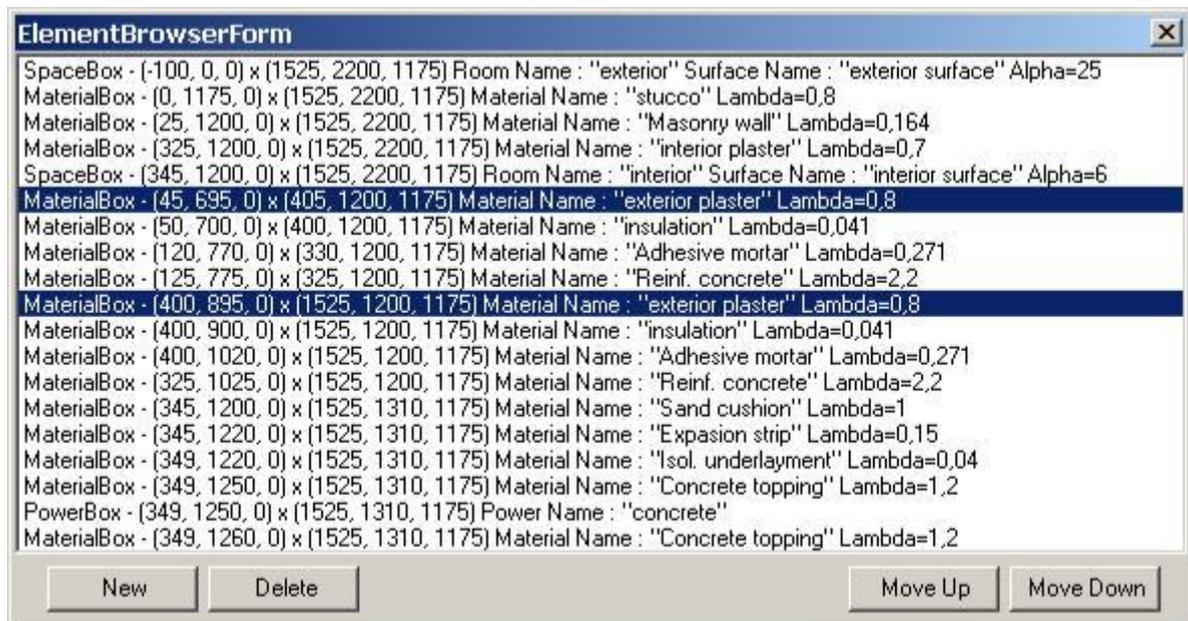
The geometric and material data of all the elements in layer 1 is hereby copied into the currently displayed second layer.

Now edit the name and thickness of this layer by moving the cursor to the name field (press TAB):

Layer 2: Floor/stucco Depth.:1

The section through the stucco (layer 2 of the model) is simpler than the previous one, i.e. composed of fewer material elements. The first stage of editing is to page to those copied elements which are not needed for this section and simply delete them, one by one.

Starting at the "top" of layer 2, that is, from the last element (18), page down in order to display and then delete the both "exterior plaster" elements - these are originally listed in Element Browser at positions 6 and 10.



element #	material
10	ext. plaster
6	ext. plaster

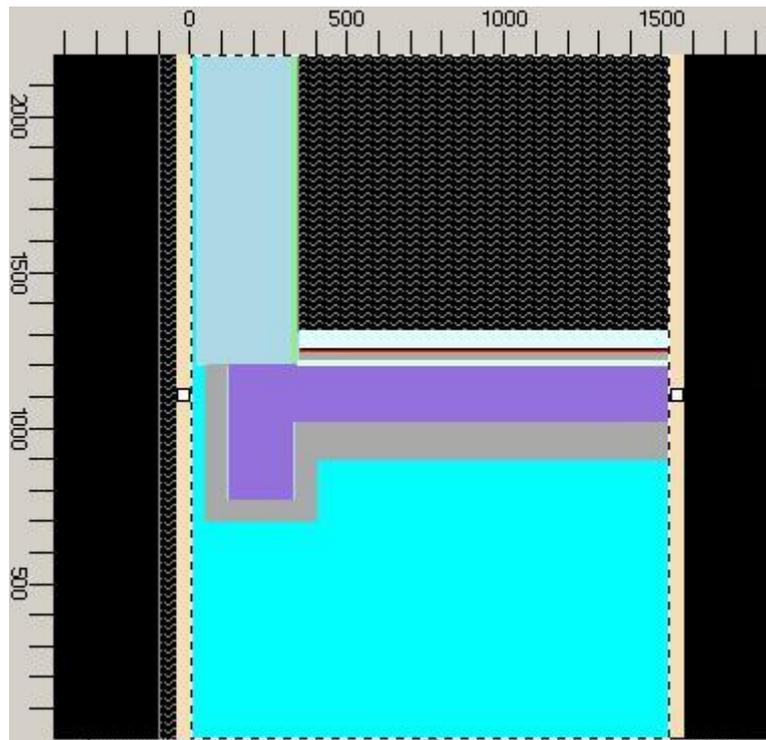
Select the Element by clicking on it and press *Delete* command button. Delete only those two elements!

The second stage of component group editing is to make any necessary adjustments in the input values of the remaining elements.

For this layer of the model, only the size of one element needs to be changed. Select element 2 - stucco and by double clicking on it open Element Editor, move the cursor to input of Y_1 (using TAB key), and enter:

X_1: 0
X_2: 1525
Y_1: 0
Y_2: 2200

Press TAB to confirm the change and re-draw the component. It should look like this:



Now the same copy and editing procedure shall be repeated to establish the next layer of the model, component layer 3:

Right-click the second layer within the Layer Browser and select *Duplicate*.

within Layer-Editor provide input:

```
Layer 3: Exp.strip layer    Depth.:4
```

Within Element Browser form delete following elements:

element #	material
17	concrete topping
16	heat source over concrete topping
15	concrete topping
14	isol. underlayment
10	adhesive mortar
9	insulation
8	reinf. concrete
7	adhesive mortar
6	insulation

Edit element 6 (formerly 11) - reinf. concrete

```
X_1: 125
X_2: 1525
Y_1: 1025
Y_2: 1200
```

The cross-section for this and several of the following model layers shows the perimeter brick as well as the insulation along the edge of the floor slab. For this part of the construction, two new elements must be inserted into the component group "under" the element of reinforced concrete.

Select "Reinf. concrete" in Element Browser. Right-Click on it and select "Add→Before Selected". Edit the input Data:

```
X_1: 85
X_2: 1525
Y_1: 1025
Y_2: 1200
Type: Material Box
```

Double-Click over the Lambda field to reveal the list of Materials. Select *insulation* and drag it onto Element Editor - drop it there assigning properties of insulation to the newly create element.

```
Material Name: insulation
Lambda: 0.041
```

The new element 6 - insulation - is generated upon confirmation (TAB), and the reinforced concrete element is now number 7.

The perimeter brick element shall be inserted "under" the insulation. Select "insulation" in Element Browser. Right-Click on it and select "Add→Before Selected". Edit the input Data:

```
X_1: 25
X_2: 1525
Y_1: 1025
Y_2: 1200
Type: Material Box
Material Name: Perimeter brick
Lambda: 0.5
```

Make sure, that the sequence of elements corresponds to the following (starting from element 5 down here): ?

element #	material
5 (space)	int. transfer coeff
6	perimeter brick
7	insulation
8	reinf. concrete

Layer 3 is now complete. Duplicate that last layer to the fourth now.

The rest of the layers of the demonstration object require editing of copied groups only with respect to name and thickness, followed by a successive deletion of elements:

```
Layer 4: Int.plaster      Depth.:20
```

Delete following elements:

element #	material
10	expansion strip
9	sand cushion
5 (space)	int. transfer coeff

Duplicate the last layer to the fifth now.

```
Layer 5: Masonry layer    Thick.:200
```

Delete:

element #	material
4	interior plaster

- Edit element 3 - Masonry wall

```
X 1: 25
X 2: 1525
Y 1: 0
Y 2: 2200
```

Layer 6 (via Duplicate of layer 5):

Layer 6: Slab perim.insul. Thick.:40

Delete:

element #	material
6	reinf. concrete

Layer 7 (via Duplicate of layer 6):

Layer 7: Perimeter brick Thick.:60

Delete:

element #	material
5	insulatio n

Layer 8:

Layer 8: Stucco layer Thick.:25

Delete:

element #	material
4	perimeter brick
3	masonry

Layer 9:

Layer 9: Ext.space Thick.:100

Delete:

element #	material
2	stucco

The construction is now complete; the eight component layers can be paged through using the Layer Browser. Save the model data as a construction file in the usual manner:

File→*Save*

For model documentation, use the options from *View*→*Evaluation&Reports*→*Data Entry Report*.

> Continue reading with "[Evaluation and Output](#)"...

Evaluation and Output in Three Dimensions

Activate the *main menu Results...* option.

You will be asked to save the project data and to provide fine-grid and solver parameters. Confirm all the three dialogs with Ok.

Numeric Results

As soon as the calculation of base solutions has been terminated successfully, the Conductance-Matrix-Report is generated and shown. Also a Boundary-Condition window is revealed to receive your input for further evaluation steps - covered just below.

Anzahl der bilanzierten Zellen: 69129

Thermische Leitwerte [W / K]			
Raum\Raum	Room 0	Room 1	
Room 0		1,917153	
Room 1	1,917154		

Verteilungsschlüssel [1]			
Wärmeq.\Raum	Room 0	Room 1	
PS 0	0,111552	0,888448	

Genauigkeitsangaben			
	Schliessfehler [W / K]	Leitwert Summe [W / K]	Leitwertbezogener Schliessfehler
Room 0	8,71675e-007	1,917150	4,54671e-007
Room 1	-8,71675e-007	1,917150	-4,54671e-007

	Schliessfehler [1]
PS 0	-3,74157e-008

(Here again, the conductance matrix would not be symmetrical enough for some purposes; in which case, the base solutions would have to be further calculated.)

These results imply that in the corner, the heat source would lose approximately 11% of its output to the exterior space (compared to 8% for the floor regions modelled two-dimensionally).

By entering same [boundary conditions as for the two dimensional](#) modelling earlier

BoundaryConditionsForm		
	Name	Value Unit
	Room 0	-10 °C
	Room 1	20 °C
▶	PS 0	6000 W/m²

the Evaluation results option provides:

Number of evaluated cells: 69129 (Nodes = 576951)

Boundary conditions and resulting Surface Temperatures / Condensing Humidity

	Air temperature [°C]	min.temperature [°C]	max.temperature [°C]	Condensing.H. [%]	f_{Rsi}^*
Room 0	-10,00	-10,00	-6,54	100,00 %	
Room 1	20,00	14,00	27,43	68,37 %	0,70
	Power [W]	Power density [W/m ²]	Volume [m ³]		
PS 0	82,98	6000,00	0,01382976		

Weighting factors for coldest surface point of each room

	Room 0	Room 1
g(PS 0)	0,000000	0,034897
g(Room 0)	1,000000	0,296522
g(Room 1)	0,000000	0,703478

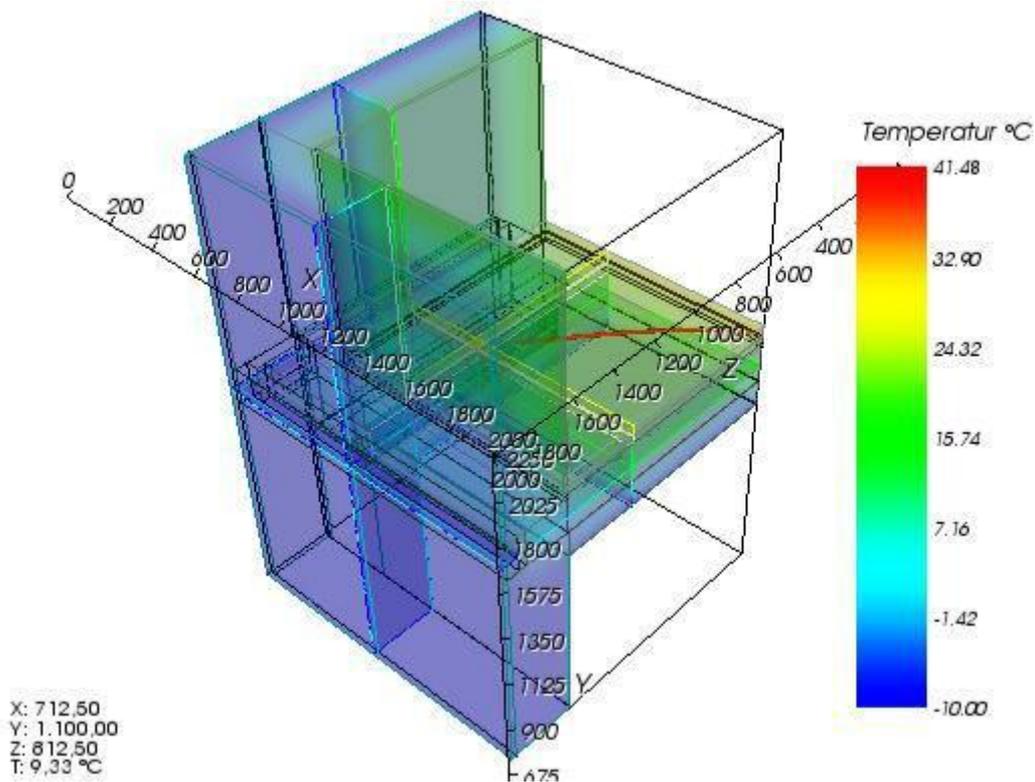
Coordinates (x,y,z) for coldest surface point of each room

	x	y	z	Temp.[°C]	f_{Rsi}^*
Room 0	0,0000	0,0000	1175,0000	-10,00	
Room 1	345,0000	1310,0000	1180,0000	14,00	0,70

Graphic Results

A picture of the overall temperature distribution within the component can be obtained by generating isotherm diagrams for characteristic cross-sections of the model. For this object, these are the x/y and z/y sections (see model sketches).

Within Results 3D View adjust the position of the rendering by rotating it appropriately to your need. You will eventually have to turn perspective view on within the 3D panel.



Now turn off the unneeded elements of the rendering by:

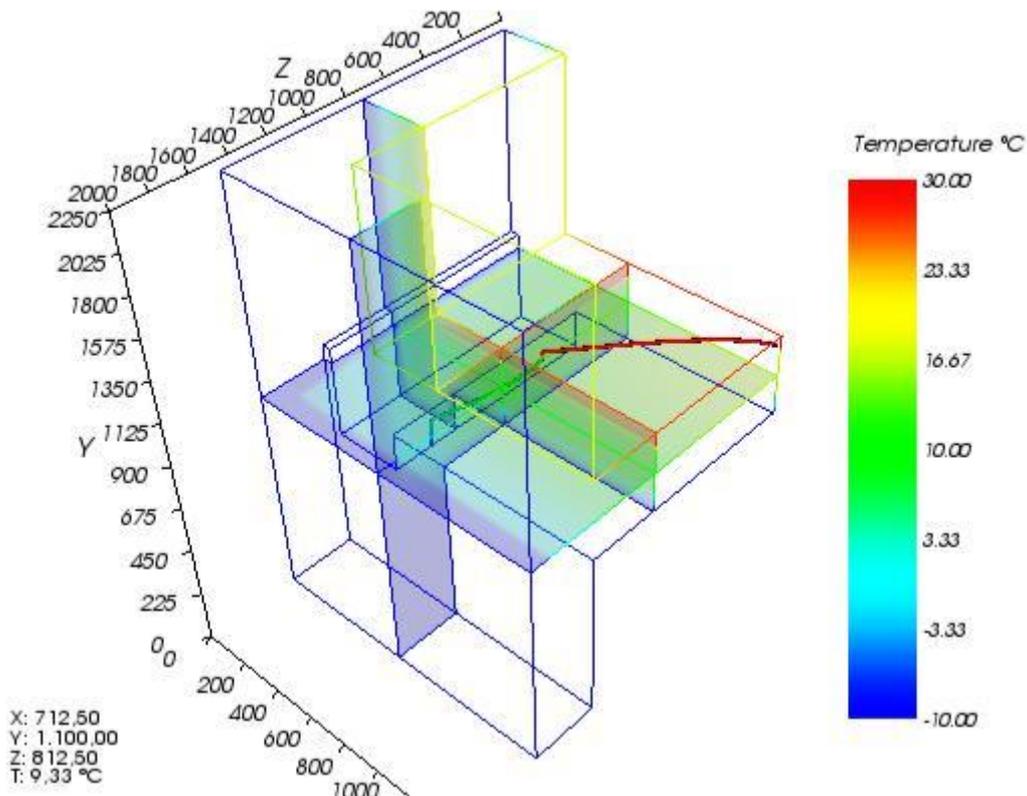
- deactivating the outline (on general panel)
- deactivating model and model slices (on model panel)
- deactivating surface (on surface panel)

The view might be partly obscured by 3d-axes. Let them be drawn "at outer edges" on Axes panel.

To see the model boundaries (edges) turn them on (on surface panel)

Adjust colorizing range of the colorbar to 30 °C for example.

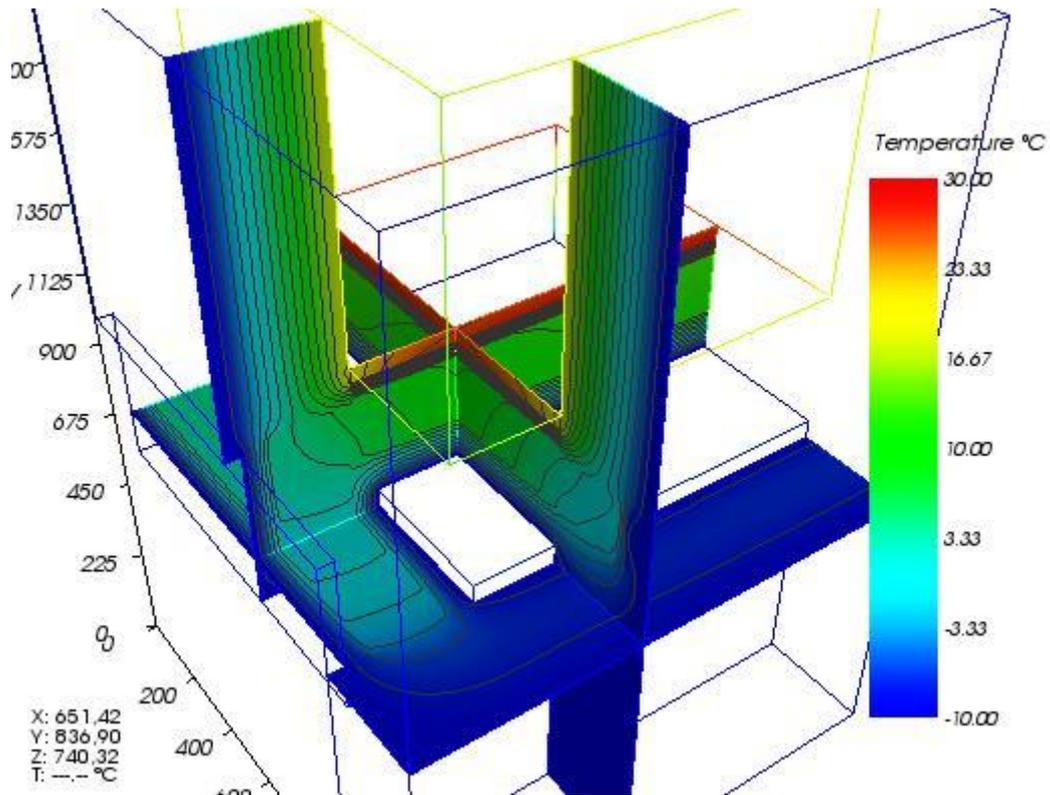
Now experiment by moving Slices X/Y/Z to emphasize the temperature distribution within the component. Right-clicking on the corresponding panel of SLice X, Y or Z exposes a context menu listing all coordinates where surfaces of the model exist. Selecting them from this context menus move the particular slice exactly to this coordinate.



By changing the opacity of the Slices X, Y and Z you can produce more understandable results. Turn them all to fully opaque.

To view Isolines, turn them on, select step of 2 °C and turn colorizing to black on isoline panel. On Labels panel turn labelling off.

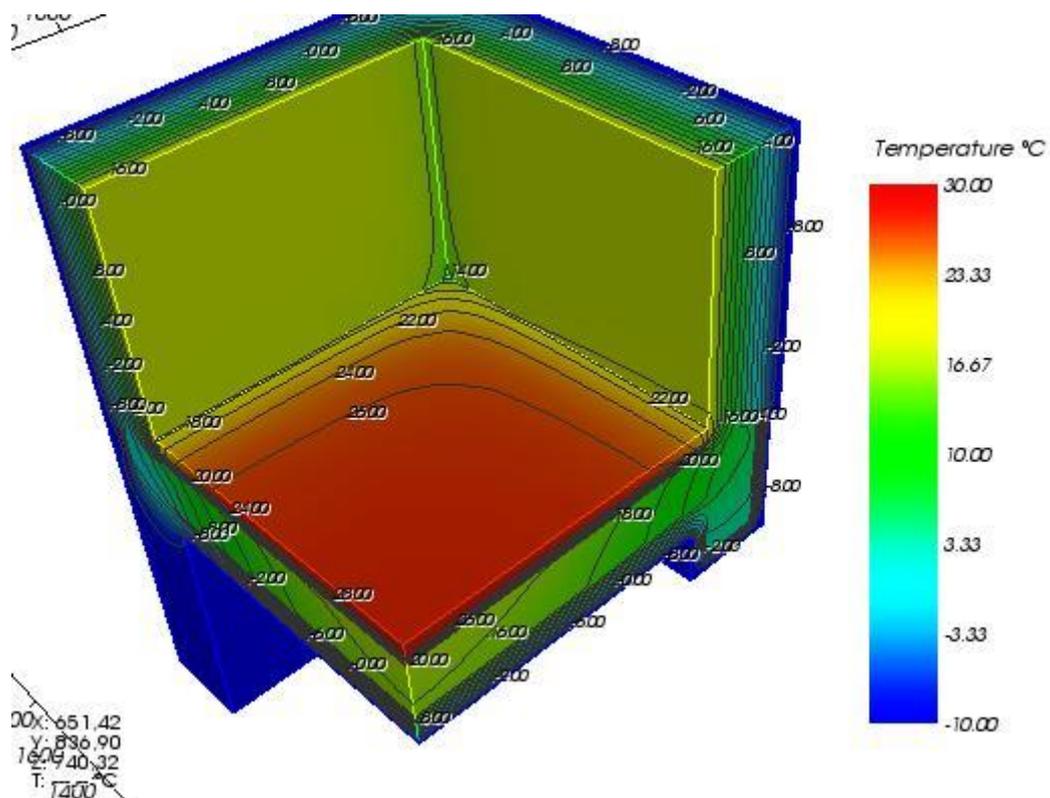
Continue rotating, scaling and panning the view according to your needs



Showing the temperature distribution on the surface can be archived by turning the surface view on and making it fully opaque (on Surface panel).

Turn Slices X/Y/Z off since their rendering are fully obscured now by the model surface and thus hidden.

Turning labelling on (Labels panel) gives more value information to the rendering. Adjust label proximity to 150 and set the "distance along line" to 1000 for reducing eventual cluttering of labels.



VAPOUR - Tutorial

Water Vapour – Vapour Diffusion, Water Vapour Bridges

Mit AnTherm mit der DAMPF-Option ist die Analyse des Wasserdampf-Diffusionsverhaltens in den Bauteilen möglich!

AnTherm verfügt über die Möglichkeiten zur Beurteilung des Kondensationsrisikos im Bauteilinneren – in 2- und auch in 3-dimensionalen Aufbauten!

Durch die Differenzbildung von Sättigung- und des berechneten Partialdruckes im Bauteil kann die Begrenzungsfläche (bzw. Linie) der vom Kondensat gefährdeten Bereiche ermittelt werden.

Zudem besteht die Möglichkeit den Weg des Wasserdampf-Diffusionsstromes als Stromlinie (ähnlich des Wärmestromes) graphisch zu verfolgen und damit die Schwachstellen der Konstruktion auch in 3-dimensionalen Fällen zu finden (die Dampfdiffusionsbrücken).

Die Vorgangsweise

Zusätzlich zu der Angabe der Wärmeleitfähigkeit der Baustoffe (Lambda) ist die Wasserdampfdiffusionszahl „Mü“ anzugeben (wie Sie dieses effizient auch nachträglich machen können lesen Sie unter „Ändern von Baustoffeigenschaften mehrerer Elemente“ auf Seite 13.

Fehlen diese Angaben, wird das Programm Sie darauf vor der Berechnung der Ergebnisse aufmerksam machen.

Die Durchführung der **Dampfdiffusionsberechnung** wird in den Solver-Parametern (Menü Bearbeiten) angefordert (im Normalfall ist diese Berechnung abgeschaltet)

Haben Sie die Mü-Zahlen eingegeben, aber die Dampfdiffusionslösung nicht aufgefordert, wird das Programm Sie auf diese Möglichkeit ebenfalls erinnern.

Achtung: Gegebenenfalls prüfen Sie in den Programmeinstellungen (Menü Bearbeiten->Einstellungen) welche der Auswertefunktionen aktiv sind. Im Punkt „Sekundärfunktionen Welche?“ sind möglicherweise einige abgeschaltet, welche sie jetzt einschalten sollten!

Vor allem sind

- Partialdruck,
- Druckdifferenz,
- Diffusionsstromdichte und gegebenenfalls auch –Vektor

jetzt von Interesse.

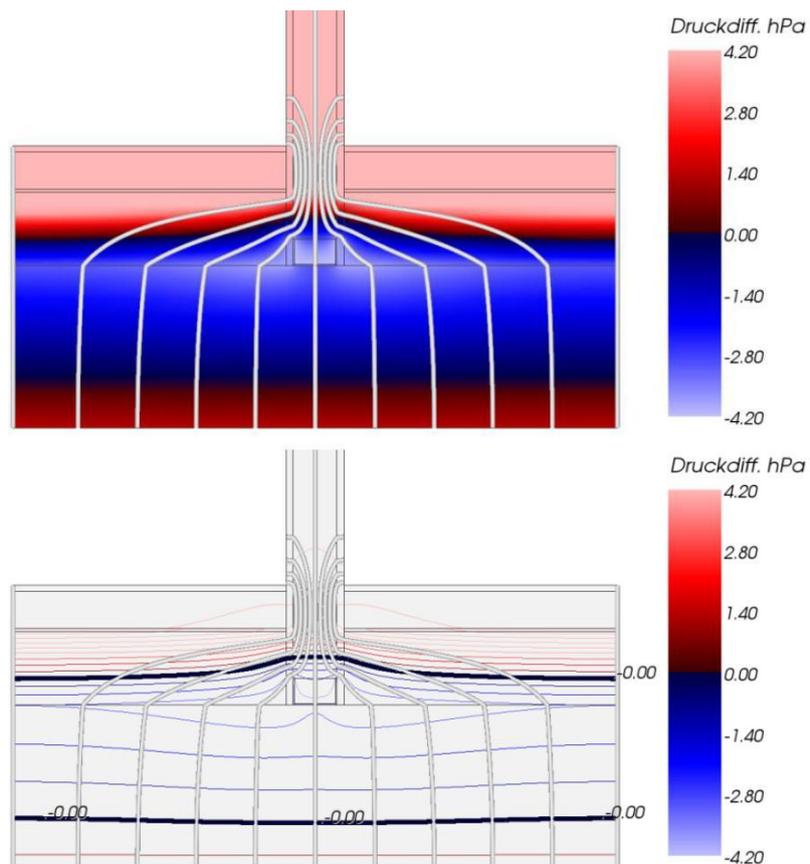
Im Leitwertbericht sehen Sie zusätzlich zu den Genauigkeitsangaben der Wärmeströmungsberechnung (Schließfehler) auch die entsprechenden Angaben der Dampfdiffusionsberechnung (soll $\ll 10e-4$)

Als Randbedingungen geben Sie neben den Temperaturen auch die relativen Raumluftfeuchten (in %) ein.

Erst in der graphischen Auswertung der Wasserdampfdruckdifferenz bekommen Sie die Aussagen über die (überschätzte) Kondensationsgefahr:

- Graphische Auswertungen anfordern (auch via Menü Ansicht->Evaluierung&Auswertungen->Ergebnis3D)
- Im Allgemein als Auswertefunktion „Druckdifferenz“ wählen – bereits hier sehen Sie den Werteintervall welches der Farbtabelle zugewiesen wurde – liegt der Randwert der Druckdifferenz unterhalb von 0, dann Kondensationsrisiko!
- Im 3D-Fall Isofläche mit Wert 0 Visualisieren (sollte es eine geben, schließt diese den Kondensationsbereich ein!)
- Im 2D-Fall die Oberfläche Visualisieren
oder eine Schnittfläche Z bei 500
oder Model mit Isolinie bei 0 (und 0-Intervalle) sichten (sollte es eine geben, schließt diese den Kondensationsbereich ein!)

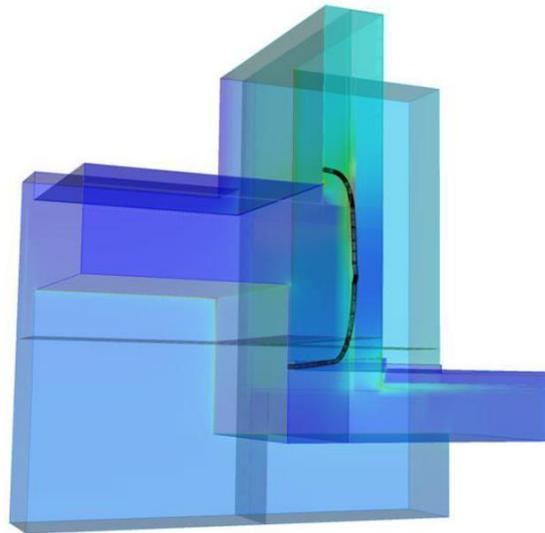
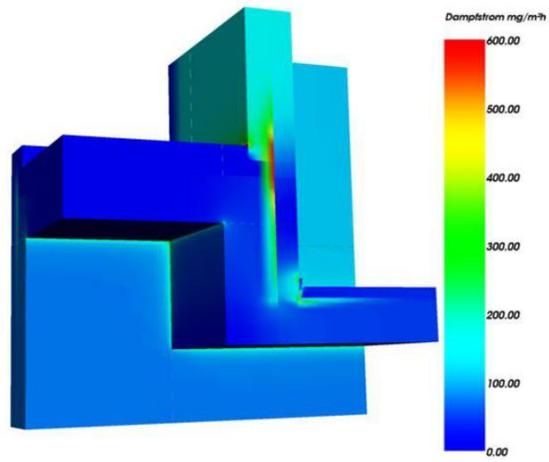
Mit Probepunkten und dann im Probepunkteteil des Ergebnisberichtes werden neben den Temperaturen auch die Druckdifferenzwerte ausgegeben.



Anmerkung: Wir haben derzeit mit Absicht keine „automatisierte“ Auswertung der Fragestellung „Kondensatgefahr“, da es vom Interesse ist nicht nur diese aber auch die Ortinformation zu erhalten – deswegen geht es nur über die Visualisierung und gegebenenfalls die Probepunkte.

Es geht aber noch weiter!

- Schalten Sie die Oberflächensicht aus, Legen Sie ein Schnitt X/Y/Z im Bauteil (im 2D-Fall soll Z=500 sein) und Schalten Sie die Stromliniendarstellung ein – es wird die Wärmestromlinie dargestellt. Jetzt ändern Sie im Allgemein die Vektorfunktion auf „Dampfvektor“ (*) und betrachten Sie die Wasserdampfdiffusionsstromlinie – diese nimmt einen vollkommen anderen weg!
- Ändern Sie in Allgemein die Funktion auf Dampfdiffusionsstromdichte und betrachten Sie mit einer Isofläche (oder Isolinie bzw. Farbendarstellung) die Wasserdampfdiffusionsbrücken bei den hohen Werten des Dampfdiffusionsstromes – ähnlich der Suche nach Wärmebrücken finden Sie so die Schwachstellen der Konstruktion im Bezug auf Wasserdampfdiffusion.



See also: [Multidimensional Vapour Diffusion, Multidimensional vapour diffusion processes](#)

Usage Hints Introductory Tutorial

Following pages collect important introductory instructions about the use of the application in the form of a cookbook.

Use [examples included in the installation](#) to become acquainted with the software. As soon as you get comfortable with the handling of the software you shall start easily with your own examples of course. You can also enter model data from scratch or transfer other projects (e.g. WAEBRU file) into AnTherm (by using the [import](#)).

[Continue reading on Starting the Program and its Windows...](#)

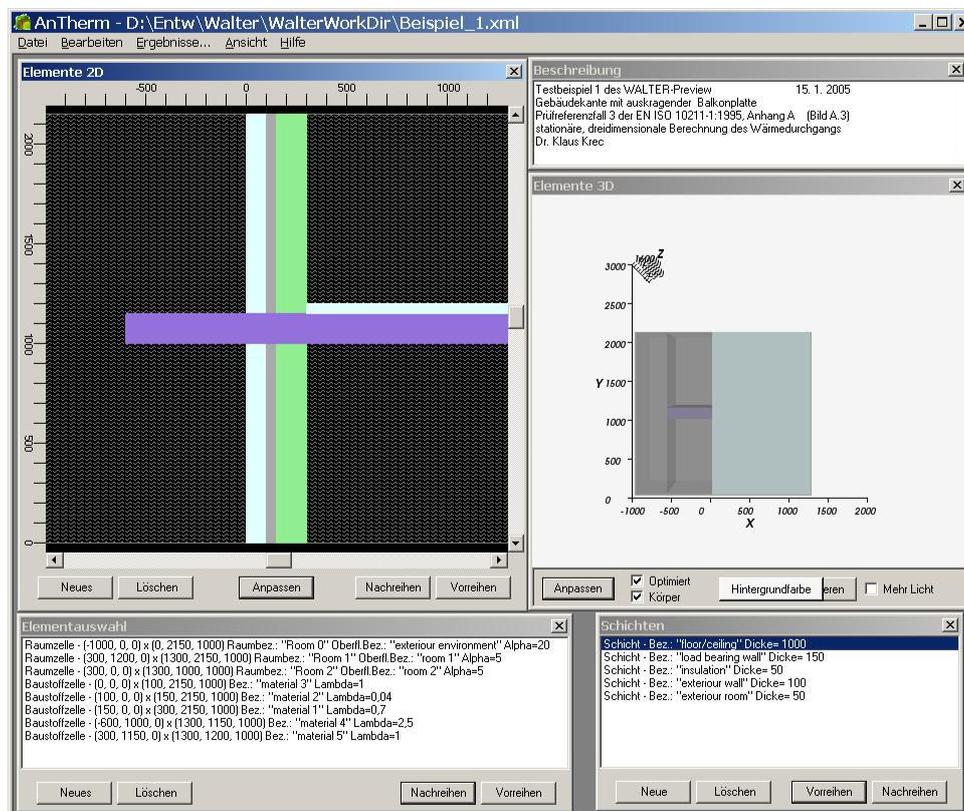
Hint: The application help window will always display on top (in front of) all other application windows when invoked directly from the application (via F1 key or the Help menu) and eventually overlap them. To have help window be freely switchable via desktop manager (which will allow you to bring the application windows to front) you shall start the application help from the windows start menu instead (Start-All Programs-AnTherm-Help) and browse the contents manually to the chapter of your choice.

With the [application setting "Help Window shown TopMost"](#) the user can decide if the Help window will be shown topmost to all application windows and automatically minimize with the application (which is the windows default behaviour) or it can be set behind the application window via typical desktop window manager (default application setting).

Starting the application - main windows

Start the application by navigating within windows menu *Start*→*Programs*→*AnTherm*→*AnTherm*.

If the [application's installation](#) has been completed successfully, the [main window](#) is shown with further work windows embedded in it.



All windows of *AnTherm* are shown on screen exactly at same positions as they were saved at earlier execution of the application.

Windows initially shown and their initial positions are preset according to [the template](#) included in the installation. All windows can be moved freely. This allows the user to arrange the screen area according to his needs.

Remark: The *graphical user interface* (GUI) of the application is initially presented in the *Language* set for the operating system (Standard/Default Language). This can be changed. Use the menu Edit→Settings... (or Bearbeiten→Einstellungen...) and within the [setting dialog](#) change the [User Interface Language](#) (or *Sprache der Benutzeroberfläche*). If the language requested cannot be provided by the program (current distribution contains English and German), AnTherm will show in English.

Position the windows and adjust their size according to your needs.	The application saves window positions into settings file AnTherm.exe.Settings.xml . Deleting this file will reset all application settings to its initial state - including window positions. We have provided some template settings with the installation.
During application's start-up the template project file projekt.xml has been loaded automatically.	If there is a file named projekt.xml within the applications installation folder it will be loaded automatically. You can use this file as a template project, containing some standard text and material definitions available immediately in the program. This behaviour can be changed by within application's settings, by requesting automatic load of latest project edited.
The main window of the application contains the menu bar of all application's functions. If required, move the menu bar to the position on the screen not obscured by other windows.	Via the menu File you create new or load existing projects and save them to project files - and you exit the program form this menu. The menu function File→Import... provides import function of WAEBRU.?BT components or the import function from DXF files and from other supported data formats. The menu Results... will lead you directly to results for the current project – to several reports and 3-dimensional graphical evaluations. If there are no results available yet the calculation is performed automatically.
For a new project you decide if it shall be 2-dimensional or 3-dimensional.	With the menu File→New you create new projects. Within the creation step you define the project type by deciding if it will be a 2-dimensional or 3-dimensional model and calculation.

[continue ...](#)

Loading a component, primary windows

<p>Select the menu Help->Example Load... and select one of the examples included in the example folder created during installation. Following the selection press <i>Ok</i> to load the project's data.</p>	<p>The application saves projects as XML files. Such files can be viewed within any standardised parser or even within browser, thus allowing processing of project's data even outside of the application.</p> <p>Remark: To quickly load examples included with the application's use the menu Help->Example Load...</p>
<p>Most important data of the project immediately loaded and its graphical representations will be shown.</p>	
<p>The window Description contains the short description of the project</p>	<p>The <i>project's description</i> is displayed within headings of all project's reports.</p> <p>You can change the <i>project's description</i> arbitrarily within the window Description</p>
<p>The Elements-2D window displays the 2-dimensional slice through the modelled construction.</p>	<p>Instructions required to change the view and on how to efficiently work with the mouse within this window are available in the next chapter of this tutorial called „further instructions“</p>
<p>The Elements-3D window display a 3-dimensional view of the modelled construction.</p>	<p>Instructions required to adjust the 3D view according to your needs are available in the next chapter of this tutorial called „further instructions“</p>
<p>The Element selection (browser) window lists all elements of the modelled construction.</p>	<p>From that list you can select elements of the construction for detailed editing, add new elements or delete obsolete ones, etc..</p>
<p>Within the Elements 3D window press the button „Copy Image“</p>	<p>The 3D view will be created and copied to <i>clipboard</i> of the operating system. During the creation of the image the cursor changes to a clock symbol.</p> <p>The image can be <i>pasted from the clipboard</i> into any suitable application for further processing, e.g. integration of an image within your personalized report.</p> <p>The image placed on the clipboard will be always created over white background to make it suitable for printing.</p>

[continue ...](#)

Calculation and Results

Click the main menu item Results and confirm with <i>Ok</i> the Fine-Grid-Parameters followed by Solver-Parameters and save project's data to a project file naming it arbitrarily.	You will be automatically alerted to save project's data to a file prior to simulation start to avoid any accidental data loss. If the request to save data is always shown can be set within application settings. Within the project's data also parameters of the fine grid and precision setting for the solver are saved.
The Solver window lets you observe the progress of the simulation process.	The animation shown at the top edge of the window informs you, that the application is currently calculating a new solution. Further messages issued by the computational part of the application shall be also shown in this window. If there are simulation results available from an earlier calculation, they will be compared to the project data just saved. If no changes of the input are detected the calculation step will be skipped..
Soon after the calculation is finished the matrix of thermal coupling coefficients and precision information for that calculation are shown.	The report window displays characteristic values of the construction in the form of the matrix of thermal coupling coefficients (L2D or L3D). The precision results shall be checked and evaluated against conformance to standardised criteria the relative close-up error < 10-4). If this criteria is not satisfied the computation shall be repeated with different parameters (of a fine grid or the solver). Thermal coupling coefficients shown might be immediately used to assess thermal energy performance of the construction just calculated (e.g. to determine the Ψ (psi) or X (chi) values for energy rating evaluation, energy pass or energy demand calculation).
The Boundary Conditions window lists all spaces for which temperatures (the boundary conditions) can be entered.	Temperatures of spaces will be pre-assigned automatically: <ul style="list-style-type: none"> • first space listed receives air temperature of -10°C • all other spaces are pre-assigned the temperature of +20°C. Spaces are shown sorted alphabetically by their names. By using some consistent naming conventions the pre-assignment might match your requirements in most cases: <ul style="list-style-type: none"> • <i>Exterior</i> is listed before <i>Interior</i> • <i>Space 00</i> prior to <i>Space 03</i> and prior to <i>Space 10</i> • <i>Room 11</i> is listed prior to <i>Room 1</i>. If the model contains power sources (heat sources) also, they are listed in the window to allow the entry of their power density [W/m ³].
To confirm the values of temperatures entered in the Boundary Conditions window press the button <i>Apply</i>	Boundary conditions entered are applied onto basic solutions (the g- values). The resulting temperature distribution within the construction is calculated as a superposition of basic solutions multiplied by the corresponding boundary condition value.
Actual results are calculated and displayed within several evaluation windows .	Results are formatted as printable textual reports which can be also saved as Adobe PDF, MS-Word or MS-Excel files for further processing. The temperature distribution is available as starting point for various graphical evaluations .
The Coupling Coefficients and Precision report and the Results report are shown as standard reports automatically.	Via the menu View→Evaluation & Reports you shall request further available reports.

[continue ...](#)

Evaluation of results

Within the Results report window click with the mouse onto the arrow symbol adjacent to magnifying glass and select the zoom factor of the report view	By default the application attempts to show the full report page within the window. You can adjust the magnification factor of that view.
Press the printer symbol and confirm various printing parameters to send the report to the printer for hard copy output.	Printing of a report can be requested via the menu File→Print also. Reports can be saved as Adobe PDF, MS-Word or MS-Excel documents for further processing.
Select the menu View→Windows→Coupling Coeff. Report.	The matrix of thermal coupling coefficients and precision output is shown.
Select the menu View→Windows→Results 3D.	The Results 3D window display the three dimensional graphical evaluation of temperature distribution and of further secondary functions (heat flux, relative surface humidity etc.) Visualizations shown can be distinguished as follows: <ul style="list-style-type: none"> • Surface temperature (Surface) • Surface of equal temperatures (Isosurface) • Distribution of temperature on a slice plane (SliceX, SliceY, SliceZ) and so on. The view can be arbitrarily moved with the mouse within the window allowing detail interrogation onto and into the calculated temperature distribution.
Within the Results 3D window rotate the view just a bit towards you (the viewer).	Likewise in Elemente 3D Fenster use the mouse to move the view by a „Track-Ball“ held under the mouse.
At the right side of the Results 3D window select Surface . Turn the switch Solid off. You will see the wire frame view onto the discretisation grid used during the simulation. Reduce the Opacity to 0.1 by using the arrows of the Opacity input field.	With the tabs Surface , Isosurface , SliceX etc. you select the tab containing parameters of respective evaluation part. <ul style="list-style-type: none"> • Solid switches between solid model view and wireframe view. • Opaque switches the transparency on and off. • Opacity defines the level of transparency between 0 (transparent) and 1 (opaque) • Active turns the visibility of the selected evaluation part on and off
Select the Isosurface and change the value of the temperature the isosurface shall express – <i>move the slider with the mouse.</i>	The isosurface connects points of equal values of temperature. By moving the <i>slider</i> you are changing the <i>isosurface</i> . The <i>colour</i> of the <i>isosurface</i> corresponds to the temperature shown on the colour scale .
Within the window Results 3D click onto the button „ Copy Image “	The 3D view will be created on the <i>clipboard</i> of the operating system. The image can be then <i>pasted from the clipboard</i> into any suitable application (e.g. Word) for example for integration into personalized report. The image placed on the clipboard is always drawn over white background for proper hardcopy output.
Within the window Results 3D select the tab General and change the primary function chosen.	If there were secondary functions calculated (this is decided by the switch „Secondary functions“ of "General" panel) a set of them can be chosen as the primary function for 3-dimensional visualization: <ul style="list-style-type: none"> • temperature °C • heat flux W/m² (the modulus and the vector field) • relative surface humidity % (only at component's surface)

	etc.
Close the window <i>Results 3D</i> by clicking onto the <i>X-symbol</i> at the top-right corner of this window.	
Close the <i>Coupling Coefficients Report</i> and <i>Results Report</i> by clicking onto the <i>X-symbol</i> at their top-right corner.	
Select File→Exit from the main menu .	If some project's data has been changed you shall save the project via the menu File→Save prior to exiting the application.

[continue with further hints and instructions ...](#)

Further hints and instructions

Elements 2D	
Move the mouse pointer over the picture shown in within the Elements 2D window. Press and hold the <i>left mouse button</i> , then move the pointer.	You can adjust the view according to your needs. The view can be „scrolled“ arbitrarily. You can use <i>scrollbars</i> to pan the visible view area also.
Press the button <i>Fit</i>	The position and scaling of the view will be adjusted in such a way, that the whole construction is made visible within the window.
Turn the mouse wheel	The view is scaled (Zoom-In/Zoom-Out). Scaling is centered around the current mouse pointer position. This allow zooming directly into the region of interest.
Elements 3D, Results 3D	
Place the mouse pointer over the view within the Elements 3D window. Press and hold the left mouse button, then move the pointer.	You can adjust the 3D view according to your needs. Moving the mouse with left button depressed results in a rotation like by turning a „Track-Ball“. The view behaves as if it were attached to a "ball" whose centre is at the centre of the view window. The mouse cursor is attached to the part of the ball that protrudes above the surface of the window.
Hold the <i>Ctrl-key</i> of the keyboard depressed, then press the and hold <i>left mouse button</i> , then move the pointer.	The 3D view is rotated around the axis in the geometrical centre of the view and perpendicular to the screen surface (rotation around the block axis , also called "Rolling"). You will need this function to flip the view "upside down" for example.
Hold the <i>Shift-key</i> of the keyboard depressed, then press the and hold <i>left mouse button</i> , then move the pointer.	The view is translated in the plane of the window following the mouse pointer,
Press and hold the right mouse button, then move the pointer.	The view is scaled (zoom-in and zoom-out). Scaling operation is executed relative to the geometrical centre of the 3d view in virtual space
Press the button <i>Fit</i>	The position and scaling of the view will be adjusted in such a way, that the whole building component fits into the view.
Fine Grid parameters	
After <i>Results...</i> (<i>in the main menu</i>) has been requested the window of <i>Fine-Grid-parameters</i> is shown.	You have the choice of two procedures: You define all the parameters, including the maximum step of the grid (<i>MaxStep</i>) You set the number of cells to be included in the equations (<i>Target model size</i>) and let the software estimate the <i>MaxStep</i> accordingly. In the letter case (<i>the model size</i> has been input) the application will iteratively adjust the <i>MaxStep</i> until the requested number of cell matches the current value +/- 10%

Streamline parameters	
<p>To receive nice streamline visualization it might be required to adjust few settings</p>	<p>The integration step length is given in millimetres. The number entered is used only at initial stage of the Runge-Kutta method implemented. Particular step lengths are adjusted adaptively - from one cell to the other. This provides the ability of very detailed visualization of streamlines!</p> <p>The maximum propagation length is given in millimetres also. During tracing a streamline the trace length is cumulated (the sum of all segment lengths of a streamline traced) and if the limit set here is reached, tracing of that streamline aborts.</p> <p>The next parameter which limits the length of a streamline is given by the maximum number of steps. The limit of 100000 segments is hard coded within the application. Higher number would overwhelm the capacity of today's hardware equipment when combined with tube visualization.</p> <p>The tube radius is given in millimetres. The initial value of the tube radius is computed based on median extent of current component (1/200 of it). Setting the tube radius = 0 will show it as simple line.</p> <p>Start at – Probe X/Y/Z: Only one streamline passing the point of intersection of the three slice planes X/Y/Z will be created. To get the streamline drawn all three slice planes must intersect within the model's body.</p> <p>Start at – Space Boundary: Produces a set of streamlines started at selected space boundary. The application calculates the sum of stream flowing to the space and the sum of stream leaving the space through the boundary (for exterior space, case with two spaces without powers sources one of this values is always 0). The sum with the largest modulus is divided into the number of intervals entered. Along the space boundary the application repeats the calculation of the stream function and places start points for the streamline at interval boundary values calculated earlier (interpolated).</p>
<p>Streamlines at equidistant intervals of heat stream from space boundary in 2D case</p>	<p>Create or load a 2D project; <u>Results:</u> Apply boundary conditions -> Graphical evaluation (Results 3D window) All streamlines will start (and in 2D case will fall into) the Z slice plane, therefore the slice Z must be active (even if transparent and thus invisible). Recommended combination of settings (allows quick initial visualization):</p> <ul style="list-style-type: none"> • General: Perspective OFF (it is 2D only), Outline OFF • Model: Model active OFF, Model Slices active OFF (later you eventually can vary the visibility of model slices at Z plane, tube radius of a streamline and colouring) • Surface: Surface active OFF (otherwise it would obscure the interior, including streamlines!), Edges OFF, Slice Edges ON (or OFF), Colorize Slice Edges ON/BLACK/WHITE (upon to your favour) • Slice X active OFF (required only for intersection of slice X/Y/Z, now disturbing only by the intersection cross shown in the view) • Slice Y active OFF (see above.) • Slice Z active ON, Value 500; Opaque ON, Solid ON, Colorizing ON (later you can vary Opaque/Solid/Colorize, the value does not need to be 500 - it is arbitrary within z-homogenous/2D models) • Isolines OFF (the combination with streamline might be very interesting later, for know let it OFF) • Labels n/a (actually no preference) • Isosurface OFF • Streamlines active ON, eventually Tube-Radius=0 (memory demand), Start-At SPACE-BOUNDARY, Opaque ON, Solid ON, Colorize WHITE • Colorbar OFF (actually no preference) • Probe OFF • Axes OFF (actually no preference) • 3D: View from TOP (The middle box-button at the right column) Perspective OFF

	<p>(button not shown depressed)</p> <p>Now, within the Streamlines tab select the components edge by selecting the space by its name and set the number of intervals At initial view we advise you to turn the tube function of the streamline off (radius = 0) - the 3D representation of a tube has potentially very high complexity (millions of polygons!) and might result in very high demand on graphics memory.</p> <p>Collect Parameters (also 3D), eventually transfer it to "standard", adjust detail (right mouse button over the list of evaluation settings).</p> <p>Save the project (together with evaluation settings collected), standard setting will be saved automatically at application exit.</p>
--	---

Modelling by dynamic overlapping of elements

Finding helpful windows	
Start the application via the Start Menu→Programs→AnTherm→AnTherm.	Preset primary editing windows of the application are exposed at their positions as saved at previous application run. The file projekt.xml , if it could be found within the application's folder, will be automatically loaded.
Look at the window Element Browser . If that window is not visible, select the menu View→Primary Edit Windows or View→Windows→Elements.	By using the menu View one can <i>open or close any of the application windows</i> . View→All Edit Windows will expose all application windows – this is the quick option allowing opening of ALL windows. It will open large number of windows, therefore we advise you to use more selective alternatives instead.. View→Primary Edit Windows exposes only most important windows required for the input. Further windows, if required, will be opened within the current editing context via submenus or clicks/double clicks within primary windows. Window→Close All Windows will close all windows to cleanup the user interface – only the main application window remains open.
Select the menu Window→Close All Windows.	All windows, except the main window, will be closed.
Select File→Load (or Help→Example Load) and look for the example file overlap.antherm within the example folder (Beispiele) of the distribution. Confirm your selection with Ok.	The 3D model of the project will be shown within editing windows. Several primary windows will be opened.
Within the Elements 2D window press the button Fit. Then, place the mouse pointer over the centre of the view within this window and turn the mouse wheel until the view of the slice shown fill 70% of the window.	The display of the model is scaled (Zoom-In/Zoom-Out). Scaling is executed centred under the mouse pointer. This provides zoomed view of the region of interest or shrinking to show parts earlier outside of the window's area.
Overlapping of elements	
After the example project overlap.antherm has been loaded, it will show an example containing two elements overlapping one another. The first element, here a SpaceBox , is overlapped by the second element, here the EmptyBox .	By overlapping the elements in the order of their entry you automatically receive dynamic excavation of elements earlier in the list. The window Element Browser shows the list of elements – the order of elements within that list is significant for overlapping results! Within our example the SpaceBox is listed PRIOR to EmptyBox . Therefore the SpaceBox (Element 1) is <i>overlapped</i> by the EmptyBox (Element 2), <i>resulting in the excavation of the SpaceBox</i> .
Within the Element Browser window click (with <i>left mouse button</i>) onto the line listing the SpaceBox .	A click onto the Element line in Element Browser the particular Element is marked selected – it is shown marked within the list. The edges of selected elements are <i>shown emphasized</i> within the Elements 2D- und Elements 3D- windows.
Within Element Browser click onto EmptyBox .	Now the EmptyBox is selected and shown emphasized. The way elements are displayed within graphical views (2D, 3D) depends on <i>Element type</i> and selected <i>colour</i> , whereas: EmptyBox is visible as an edge of the „empty area“ (within the 3D view EmptyBox-es are shown and visible only if selected) SpaceBox is shown overlaid by wave lines within 2D view, and within 3D view is displayed partly transparent. The EmptyBox is used for <i>excavation</i> or to <i>create cut-off planes</i> . A metaphor of „vacuum“ <i>without any form of heat transport</i> might be helpful. A SpaceBox renders the idea of a <i>space with defined temperature</i> and the heat transport occurs only at the surfaces of modelled construction (MaterialBox-es) connected to that space.

Changing the overlapping order	
<p>Within the <i>Element Browser window</i> select the <i>EmptyBox</i> (by clicking on it) and then press the button <i>Relegate</i> to move the selected element (the <i>EmptyBox</i> in this example) one position back in the list.</p> 	<p>By changing the order of elements with the element list different overlapping effect results. Because the <i>EmptyBox</i> is now placed prior to the <i>SpaceBox</i> within the list, the <i>EmptyBox</i> is overlapped now – the <i>SpaceBox</i> is visible in its original size and not excavated.</p>
<p>Within the <i>Element Browser window</i> select the <i>EmptyBox</i> and then press the button <i>Promote</i> to move the selected element (the <i>EmptyBox</i> in our example) one position forward in the list.</p>	<p>The original state of the project shall be now restored - the first element, the <i>SpaceBox</i>, is overlapped by the second element, the <i>EmptyBox</i>.</p>

Various types of elements

<p>Within the Element Browser window double-click onto the EmptyBox to expose the Edit Window of the element (Element Editor). Position the <i>Element Editor window</i> upon to your needs.</p> <p>Remark: Select Help→Example Load and look for the example file overlap.antherm within the example folder (Beispiele) of the distribution.</p>	<p>The window <i>Element Editor</i> is used to enter and modify properties of one element selected for editing. Following properties shall be entered: geometrical placement (the <i>coordinates</i> x1,x2,y1,... Element type: EmptyBox – a deletion element MaterialBox – an element filled with material PowerBox – a range covered by a power source SpaceBox – a room (filled with air of constant temperature) optionally: an assignment to a editing group(s) Die <i>Thermal Conductivity</i> (λ) or <i>Thermal Transfer Coefficient</i> (α) optionally: colour for visualization (currently no images nor patterns can be defined) <i>Name</i> of a Space or of a Power Source</p>
<p>Within Element Editor window change the y1 coordinate of the element EmptyBox from 11 to 9. To accomplish this click onto the input field y1, delete the content by using the <i>Del and/or Backspace</i> key and enter 9.</p>	<p>If the data has not been confirmed yet by leaving the input field, the current input has no influence onto the modelled construction.</p>
<p>Confirm entered data with the TAB key.</p>	<p>The change will be visible immediately in all other windows.</p>
<p>Rotate the 3D view by 90° to the left - click onto the window <i>Elements 3D</i> and while holding the left mouse button pressed over the 3D view move the pointer to the left.</p>	<p>In our example (project file "overlap") the EmptyBox extends out of the SpaceBox – we want to change it now:</p>
<p>Click onto the <i>Element Editor window</i> (if the <i>Element Editor window</i> is obscured or hidden by other windows, <i>double-click</i> onto the EmptyBox within <i>Element Browser</i>).</p>	<p>You can reveal and activate the <i>Element Editor</i> window by: a double click onto a element within <i>Element Browser</i> via a <i>context menu Edit</i> of the <i>Element Browser</i> - the menu is shown when the right mouse button is pressed.</p>
<p>Within <i>Element Editor</i> use the TAB key to proceed to the input field z1</p>	<p>By pressing the key TAB (or Shift-TAB) you move the input focus between the input and control elements of a window – not only input fields but switches and buttons also. The exact focus transfer and usage scenarios are described within the documentation of the windows operating system.</p>
<p>For z1 enter 1, press TAB and for z2 enter 9. Press TAB.</p>	<p>The change has been applied and made visible immediately in all other windows.</p>
<p>Within the <i>Type</i> field select MaterialBox by choosing that type from the list (or make the selection with up- or down-arrow key).</p>	<p>The change has been applied and made visible immediately in all other windows. The <i>MaterialBox</i> is displayed as solid and opaque object. During the change of <i>Type</i> other input fields are renamed and activated for the input. The input fields of <i>Lambda</i> and <i>Colour</i> are active now.</p>
<p>Press TAB key to focus onto the input field <i>Material Name</i> and enter „Material XYZ“.</p>	<p>The entry of material properties (<i>Name, Thermal Conductivity, Representation Colour</i>) can be provided directly within the <i>Element Editor window</i>. Alternatively you could pick material properties from the <i>Material List</i> also – but within this tutorial we would like to focus on basic data entry methods at first.</p>
<p>With the TAB key move on to the input field <i>Lambda</i>, enter 1 and confirm the input with the TAB key.</p>	
<p>Change the <i>Colour</i> used to display the element - <i>double-click</i> onto the rectangle next to the label <i>Colour</i>:</p>	<p>The standard windows' colour editor will be exposed.</p>
<p>Select the <i>Colour</i> within the <i>Colour window</i> and confirm the choice with a click onto the <i>Ok</i> button.</p>	<p>The <i>change of the colour</i> is immediately shown within the graphical view.</p>

The Spaces (the boundary conditions)	
Within the Element browser window select the SpaceBox. Double-click onto the SpaceBox to expose the Element Editor window .	
In the input field Surface name enter „Exterior surface“.	Analogically to materials the surfaces shall have different properties too. Akin to material's Thermal Conductivity the surface is described in terms of its Thermal Transfer Coefficient.
Confirm the name entry with a TAB key and enter 10 into the input field Alpha.	
With the TAB key move to the field Space name and enter „Exterior space“ .	- one creates the list of the boundary conditions for the simulation All SpaceBoxes are consolidated by their space name to one space. Each such space will be assigned a value of temperature (the boundary conditions). With consistent assignment of space names – e.g. exterior, living room, kitchen, bath – or – e.g. Room 0, Room 1,
Confirm the entry with the TAB key	Changed data is reflected in all windows.

The order of elements - overlapping effects

Within the <i>Element Browser window</i> click <i>New</i> with the mouse.	A new, empty element will be created and appended to the list of elements. The <i>Element Editor</i> displays properties of the new element. It is an <i>EmptyBox</i> with zeroed dimensions (elements of zeroed dimensions have no effect within the model)
Following the earlier description enter following data within <i>Element Editor</i> for the new element: <ul style="list-style-type: none"> • X1: 2, X2: 8 • Y1: 2, Y2: 8 • Z1: 2, Z2: 8 • Type: <i>SpaceBox</i> • Surface name: <i>Interior surface</i> • Alpha: 10 • Space name: <i>Interior space</i> and confirm the entry with the <i>TAB</i> key.	The new element is enclosed by other elements. Due to the fact being the last element in the list it overlaps all other elements.
Within the <i>Element3D window</i> switch the visualization from <i>Solid</i> to <i>Wireframe</i> by turning the switch <i>Solid</i> off.	The wire frame representation within the 3D view allows us to see selected elements not being covered (obscured) by surfaces shown for solids.
Within <i>Element Browser</i> click onto various elements and observe the changes to the selection display in graphical windows.	
Within <i>Element Browser</i> click onto the last element – it shall be the <i>SpaceBox</i> of the „Interior space" – to select that element.	
Within the <i>Element Browser</i> click several times onto the button <i>Relegate</i> - until the <i>SpaceBox</i> of the interior space has moved to the Front of the List.	While changing the order of elements please observe the effects of overlapping within graphical views which significantly change being dependant on current sequence.
Reorder elements to receive following final sequence: <ul style="list-style-type: none"> • Exterior space • Material • Interior space to create overlapping effect resembling the modelled construction by using <i>Promote/Relegate</i> command from the <i>Element Browser</i> .	The element "Exterior space" encloses the rest, but it is overlapped (hollowed) by element following it in the list. Similarly the material element is hollowed by the interior space box.

Why is overlapping/hollowing of elements done dynamically? Where is the gouge function.

Many CAD application provide a static hollowing function (i.e. you select an element or a group of 3D objects and the application to gouge through them). Such hollowing operation result in extremely large number of oddments, normally much higher then the original number of elements. At the same time the original structure gets lost after such operation. If the user decide, that the operation had undesired effect he can either repeat the data entry or must undo the whole operation.

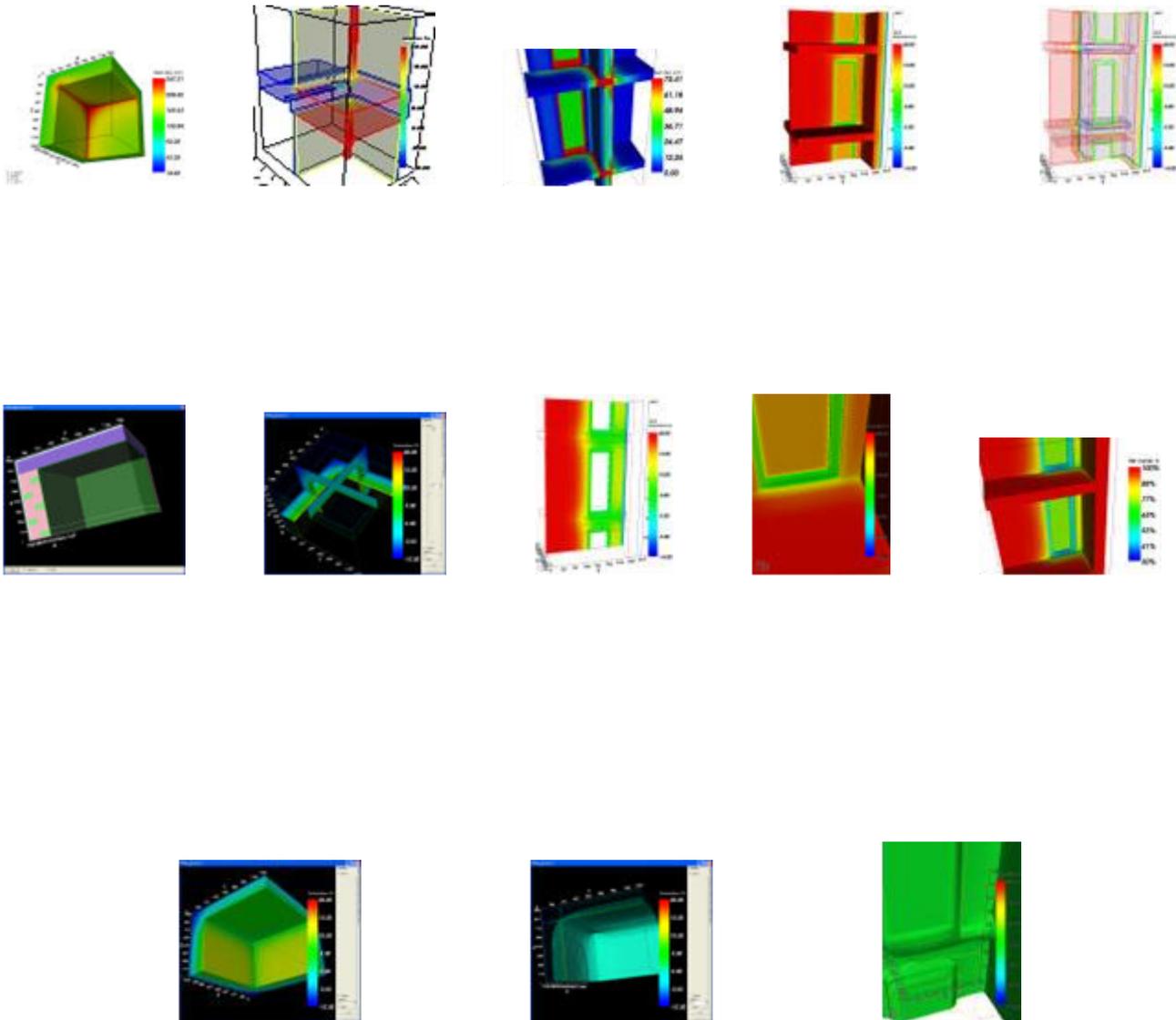
To compensate on that tree element structures are used often – each element, after being hollowed by other elements, brakes into a new branch holding all resulting oddments. As a result of many such operations a very deep tree structure is created which is very difficult to undo and, with very complex models, cannot be managed without additional, extensive and potentially very sophisticated tools.

For the both reasons (unmanageable complexity at each further step, high learning costs, error-prone programming effort) we have decided to keep things simple within a simple, linear list and use the order of the list for dynamic hollowing providing same result at much less learning effort.

[Return to Tutorial Selection...](#)

Example images

We have collected few pictures of evaluation results here. Large choice of images is [available in the web](#).



T.Kornicki, Dienstleistungen in EDV und IT
 A-1230 Wien, Othellogasse 1/RH 8/2
 Tel./Fax. +43-1-6157099
 email: tkornicki@chello.at

3/16/2005

Codename WALTER
 Preview Version (pre-Alpha)
 (c)T.Kornicki, all rights reserved

DEMO3_3D:

Wandschenkel aus Ziegelmauerwerk, beiderseitig verputzt;
 oberer Abschluss durch verputzte Betondecke.
 Dreidimensionale Berechnung:

Datei: D:\Entw\Walter\Walter\bin\Debug\demo3_3d.xml
 Anzahl der bilanzierten Zellen: 104652 (Knotenzahl > 837216)

Randbedingungen (Lufttemperaturen / Leistungen)

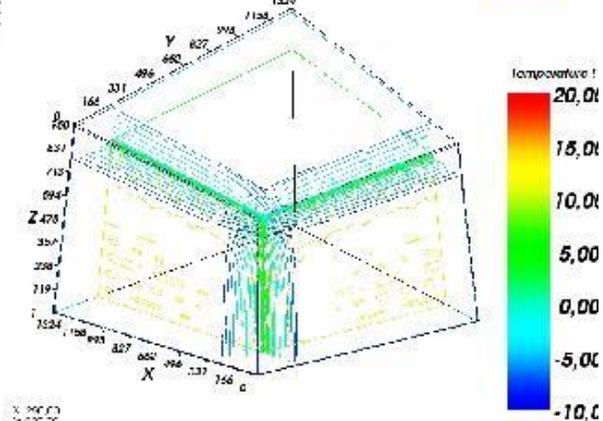
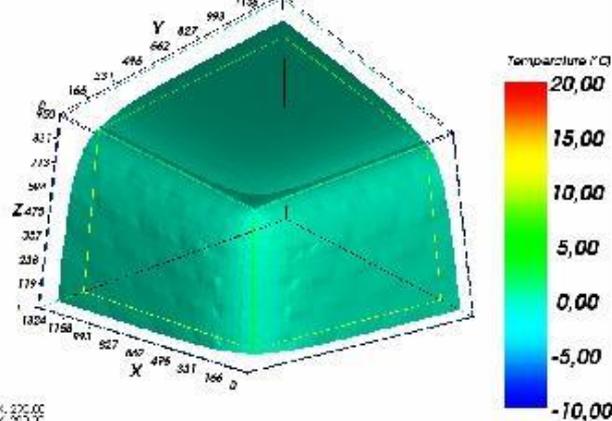
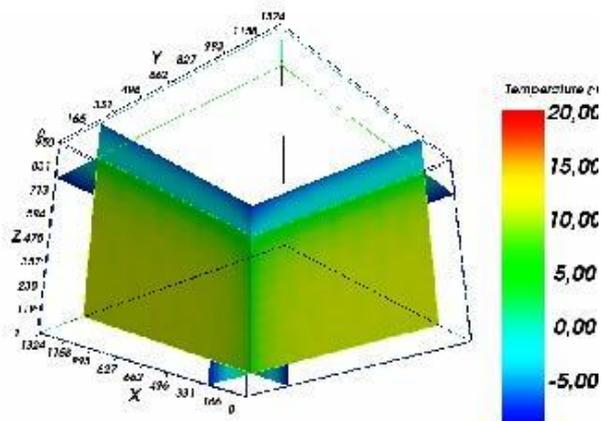
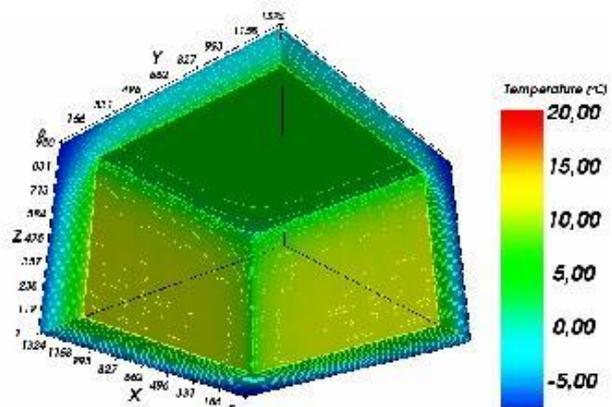
	Raumtemperatur [°C]	min. Temperatur [°C]	max. Temperatur [°C]	Grenzfeuchte [%]
Room 0	-10,00	-9,96	-5,49	%
Room 1	20,00	-1,74	11,17	22,62 %

Gewichte für den kältesten Oberflächenpunkt eines jeden Raumes

	Room 0	Room 1
g(Room 0)	0,998566	0,724551
g(Room 1)	0,001434	0,275449

Koordinaten (x,y,z) des kältesten Oberflächenpunktes eines jeden Raumes

	X	y	Z
Room 0	10,0000	10,0000	930,0000
Room 1	290,0000	290,0000	750,0000



T.Kornicki, Dienstleistungen in EDV und IT
 A-1230 Wien, Othellogasse 1/RH 8/2
 Tel./Fax. +43-1-6157099
 email: tkornicki@chello.at

3/16/2005

Codename WALTER
 Preview Version (pre-Alpha)
 (c)T.Kornicki, all rights reserved

DEMO3_3D:

Wandschenkel aus Ziegelmauerwerk, beiderseitig verputzt;
 oberer Abschluss durch verputzte Betondecke.
 Dreidimensionale Berechnung

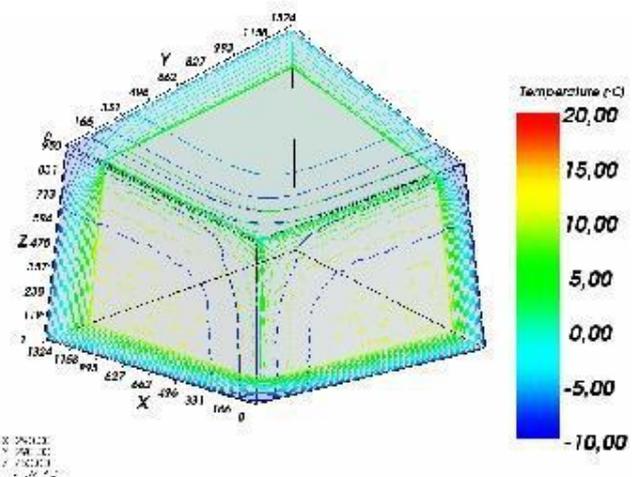
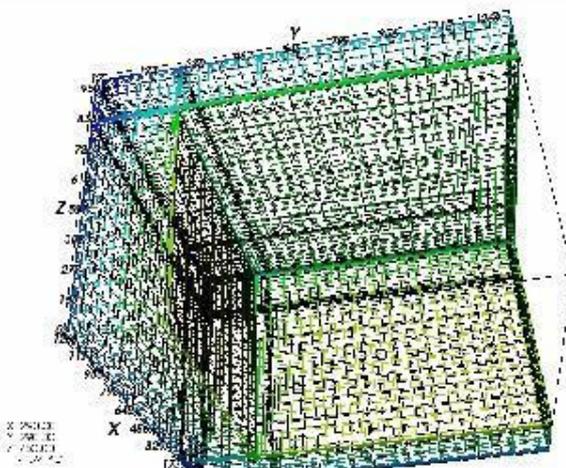
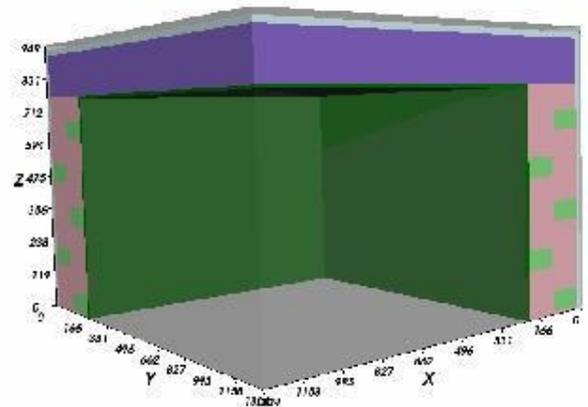
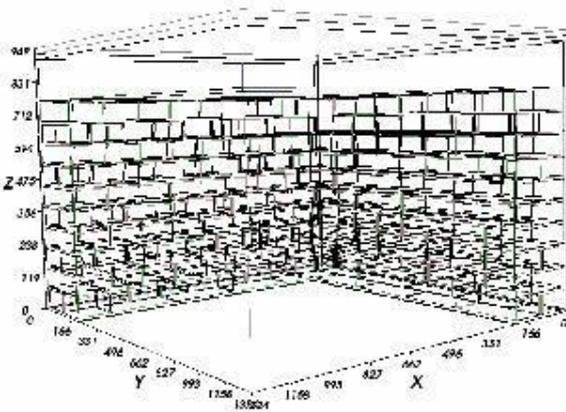
Datei: D:\Entw\Walter\Walter\bin\Debug\demo3_3d.xml
 Anzahl der bilanzierten Zellen: 104652 (Knotenzahl > 837216)

Thermische Leitwerte [W / K]

	Room 0	Room 1
Room 0		6,579532
Room 1	6,579507	

Genauigkeitsangaben

	Schliessfehler [W / K]	Leitwert Summe [W / K]	Leitwertbezogener Schliessfehler
Room 0	-2,46898e-005	6,579510	-3,75254e-006
Room 1	2,46898e-005	6,579530	3,75252e-006



FAQ - frequently asked questions

How to find and where to get the complete user documentation? Can it be downloaded?

Complete user documentation is provided electronically and can be viewed by following means:
directly within the program by choosing Main menü->Help->Help ; or

by pressing the F1-Key on the Keyboard. This functionality is provided for each application window; or
from the windows start menu. Therein choose "All programs->AnTherm" to get further option to open program
documentation, tutorial and so on.

The documentation browser (MS Help Viewer) can show table of contents and provides search functionality. There is also
possibility to page forward and backwards sequentially through the documentation. The usage of the documentation
browser shall be very intuitive and easy. There is also exhaustive documentation for this browser which can be reached via
the menu "Help on Help".

Separate documentation pages can be also printed. Printing of the whole documentation is currently not supported nor
allowed by the copyright and the license terms.

By choosing this (electronic) solution we are able to provide the most current documentation content to the user.

Up to date online version can be found at <http://help.antherm.eu/> .

How much RAM memory does the software need?

We recommend having 1 Gigabyte RAM or more installed, to be able to efficiently use versatile abilities of evaluation. We
explicitly dissuade you from extensive use of virtual memory (paging file, swap-space).

Memory requirements of AnTherm will double with VAPOR-option active (i.e. when AnTherm calculates the vapour diffusion
solution and such results are included in the evaluation also)

See also: System requirements

Which operating systems can AnTherm be used on?

AnTherm has been tested on various configuration of Windows-XP Professional, Windows Vista 32bit and 64bit, Windows 7
32bit and 64bit. It shall also properly execute on MS Windows XP Home, Windows NT 4.0SP6, Windows 2000 with possible
limitations .

Microsoft .NET must be installed prior to installing AnTherm.

This software cannot be used nor installed on Windows ME, Windows 98 or older. See also: System requirements.

Why it is only possible to show and enter rectangular elements? How to deal with sloped boundaries?

In this version you use the part of the application which does only support input of rectangular or cubic elements.
Boundaries of these elements must be parallel to coordinate axes. The limitation to work only with rectangles or cubes
originates in the computational algorithms used.

It is intended to provide sloped or curved input processing in some future version of the program. This future software
component will dissolve such sloped boundaries into stepped lines.

Currently you cannot process sloped or curved lines directly. To support the input of constructions with slopes or curvatures
however, you must resolve such figures into stepped lines by turning to functionalities of typical CAD-programs. In such case
you create DXF files which can be then imported via the CAD interface of AnTherm.



Modelling the construction component and Element Types

Modelling of the construction component is done by input of elements. Boundaries created by intersections of such model elements are automatically created.

Initial modelling space is filled with adiabatic condition. This initial condition is equivalent to an [“Empty” element type](#) of infinite dimensions. “Empty” elements can be also used to create additional adiabatic boundaries if required later in the resulting model. Intersection of [material elements](#) of a component results in creation of adiabatic boundaries thus no heat flow is possible through such boundary (for vapour diffusion an “Empty” means no vapour diffusion at the boundary at all).

Modelling of thermal boundaries is done by placing [“Space” type elements](#). Different spaces are distinguished by their names (text like “exterior”, “interior”, “room A” or “Bathroom”, “Loggia” etc.). Intersections of material elements with spaces create a surface boundary described by its thermal transfer coefficient and respective space name. Later on (during evaluation) spaces are assigned to real boundary conditions – in steady state case represented by the air temperature of that space (relative humidity is used also during vapour diffusion simulation). Varying of thermal transfer coefficient is done by entry of multiple space element with different surface property but still assigned the same space name (later assigned to temperature).

Modelling the constructions themselves is done with [“Material” elements](#) described in terms of heat conductivity (vapour diffusion number μ is also used for vapour diffusion calculation). Intersection of material elements creates material boundaries. Intersection of material with space creates surface boundaries. Intersection of material with empty creates adiabatic boundary.

There is also a term of [“Power Source” elements](#) defining extents producing or consuming heat (sources or sinks). Heat sources can only be applied to material thus telling that these regions produce/consume power. Different power sources are distinguished by their names (text like “heating coating”, “floor heating”, etc.). The interior of “Power Source” elements overlapping material create power source areas. Power sources are in that case different from other element types in that they overlap materials only, but they can still be overlapped by any other element type. Later on (during evaluation) power sources are assigned to real boundary conditions – in steady state (stationary) case represented by power density.

How to Undo mistaken editing?

Can undone action be redone?

AnTherm keeps track of the editing sequence up to 1000 editing [actions which can be easily undone](#) to revert to the input state valid prior to the respective action.

Actions Undone recently can be redone in the same order.

Regardless of this powerful ability of the program you shall save your project data regularly.

How to model decreasing transfer coefficient at corners?

In the area of decreased transfer coefficient apply additional [space element assigned to the same space name but with different surface property](#). Same applies at areas of increased transfer surfaces.

Currently there is no means of automatically generate such areas.

Best and easiest method is to copy the space of the required name, reduce its extents to the area of interest and apply the changed transfer coefficient to it (eventually also changing the name given to the surface) – but still keeping the space name unchanged.

How to create adiabatic boundary?

Initially, modelling space is automatically filled with adiabatic medium, thus any material element is placed within adiabatic boundary conditions at first. By entering space type elements you apply thermal conditions at intersections with materials. Use power source elements to mark areas of heat source/sink over materials. Placing [“empty” elements](#) creates adiabatic boundary at intersections with materials.

Can AnTherm show heat stream lines?

Yes. One can define a starting point from which the heat flow will be traced and shown as a stream line. Detailed description of this function can be found in chapters [Results 3D](#) and [Stream lines](#).

In 2-dimensional cases in particular multiple heat streamlines can be shown simultaneously.

Important: Showing vector fields requires the calculation of the particular secondary function being turned on. Please check the state of the switch "[secondary function](#)" within the "General" tab and the [name of the vector function chosen for evaluation](#) also.

If the respective function is not shown please check the [application setting "Which Secondary Functions"](#) if the right one is active also.

How to show multiple heat streamlines in equidistant intervals of the heat stream from one space boundary in 2D case?

Such an evaluation is only possible for 2D projects (actually for all such projects in which the construction is homogenous in the Z-directions and without space boundaries in this direction). You will be notified that such evaluation is possible or not within the control tab „[Streamline](#)“ of the [Results 3D window](#). If the option „Start at: Space“ is disabled then this specific evaluation is not available in the context of supplied computational results of the calculated model.

Because multiple streamlines will be drawn from (or in 2D case onto) the [Z slice plane](#), this slice plane must be [active](#) (and also [visible](#)). Following recommended [setting combination](#) can be used for quick initial display:

- [General: Perspective](#) OFF (because only 2D), Bounding box OFF
- [Model: Model](#) active OFF, Model slices: active OFF (later you can vary the visibility of model slices by varying Slice-Z, Streamline-Tube-Radius and Colour)
- [Surface: Surface](#) active OFF (otherwise the interior, incl. streamlines, is obscured!), Edges OFF, Slice-Edges ON (or OFF), Slice Edges Colouring ON/BLACK/WHITE (up to your favour)
- [Slice X](#) Active OFF (alls slices only needed for the streamline from the cross point of Slices X/Y/Z, otherwise only disturbing cross lines shown)
- [Slice Y](#) Active OFF (see above)
- [Slice Z](#) Active ON, Value 500; Opaque ON, Solid ON, Colouring ON (you can later vary Opaque/Solid/Coloured; Value need not be 500 - does not matter on Z-homogenous/2D-Modell)
- [Isolines](#) OFF (later can try the combination of isolines and streamlines resulting in very interesting pictures)
- [Labels](#) n/a (does not matter)
- [Isosurface](#) OFF
- [Streamlines](#) Active ON, ev. Tuberadius=0 (to save memory), Start-At SPACE, Opaque ON, Solid ON, Colouring WHITE
- [Colormap](#) OFF (actually don't care)
- [Probe](#) OFF
- [Axes](#) OFF (actually don't care)
- [3D: View](#) from TOP (right-middle box button) Perspective OFF (button not sunken)

Now, within the tab "Streamline" you can choose the space boundary by its name and the number of intervals It is recommended to turn the tube streamline off (by setting the radius to zero) at the beginning because the 3D representation of the tube is very complex and can cause high memory demands when drawing many millions of polygons!

Can AnTherm show heat stream by vectors / arrows?

Yes. Such visualization can be created by using the [Vectors \(Hedge-Hog\) evaluation](#).

Can AnTherm show traces of vapour transport?

Yes. One can define a starting point from which the vapour diffusion flow will be traced and shown as a stream line. Detailed description of this function can be found in chapters [Results 3D](#) and [Stream lines](#).

In 2-dimensional cases in particular multiple vapour transport streamlines can be shown simultaneously.

This evaluations are available only with the [VAPOUR-Option of AnTherm](#).

Where to find harmonic thermal coupling coefficients?**How to calculate the phase shift (time lag) due to dynamic heat capacity effects?****Can AnTherm calculate dynamic, transient heat transport problems incl. heat capacity?**

Yes. In addition to solving the steady state thermal transport problem AnTherm can solve [harmonic, periodic problems](#) for arbitrary period lengths (typically one year and/or one day).

AnTherm will output [harmonic thermal coupling coefficients](#) (also by their amplitude and the phase shift, time lag) calculated directly and independent from boundary conditions.

With [modelling periodic boundary conditions](#) (fully automated Fourier analysis) time dependant evaluations are achieved - either a 3D view at some point in time, then available to creating [time series animation](#), or [timelines of temperature changing in time](#) at selected location of the 3D model.

Subject to the TRANSIENT / HARMONIC license option.

Can AnTherm show isolines (isotherms)?

Yes. AnTherm can display [isolines](#) of temperature (the isotherms), isolines of steam density magnitude (heat or vapour flux), isolines of dew point etc. at all construction [surfaces](#). Isolines can also be displayed on [slice planes](#) cutting through the building construction.

Detailed description can be found in chapters [Results 3D](#) and [Isolines](#).

Is there any validation, benchmarks of AnTherm?

It is self-evident, that AnTherm is validated according to [EN ISO 10211:2007](#) and therefore it qualifies as three dimensional precision method of "class A" according to this standard. The validation of AnTherm as conforming to the standard [EN ISO 10077-2:2003](#) is also in place.

Furthermore, methods and algorithms implemented in AnTherm are conformant with such [standards](#) as EN ISO 6946, EN ISO 13788, EN ISO 13370 etc.

Is it possible to change the automatically assigned colouring of material elements based on the value of thermal conductivity?

Yes. By default AnTherm assigns colour to each material element based on its thermal conductance. Data is passed to the colour table which defines different colours for different intervals of thermal conductivity values. This colour table can be modified by the user. The user has to edit [default colour table LambdaToColor.ColorList](#) manually. On each change to element's thermal conductivity the program will automatically reassign the colour according to definitions provided in the above colour table.

Furthermore, the user can assign any colour to any selective element manually overriding this automatic setting. When the coloured rectangle in the [Element-Editor](#) is double clicked the user is presented a colour chooser. It shall be noticed, that such explicit assignment of colour will be lost as soon as change to the thermal conductivity of that element is applied and automatic assignment will take place again.

This default behaviour of automated colour assignment by lambda can be turned off.

See also: [Element Editor, default colour table LambdaToColor.ColorList](#)

How to change the colour gamut of the temperature field?**How to change colouring of isolines?**

Within the [evaluation window "Results 3D"](#) you will find a [control panel "Colortable"](#). Therein you can make modifications of the colour gamut. You can define the interval to be coloured and switch cropping of the chosen range. You can also choose different [Colour table](#) instead of the "Default" one.

There is preconfigured [Collection of standardized Colour tables](#) included of which one can be selected but not customized. The chosen colour table applies to all ["Evaluation functions"](#) equally - there is no means of applying different colour tables to different functions selectively. Also displaying of more than one function during evaluation (temperature, heat flux, dew point values).

See also: [Results 3D window, Colortable \(control panel\), General \(control panel\)](#)

Where to find the material catalogue?

Exhaustive function of material catalogue can be found [Material master data](#).

Material data can also be [merged from other AnTherm projects](#).

You can create your own material list and (optionally) save it as [projekt.xml](#).

I can't enter values with decimal places!

This is related to localisation settings of your operating system and your keyboard.

Please read additional information about [country specific localisation](#).

How can I create a report containing graphical evaluations and adjust it to my needs?

[Reports](#) created with the program can be saved PDF-, RTF-, XLS- or DOC-files. [3D displays](#) can be copied to the clipboard, saved as [Image file \(PNG, JPG, GIF, TIF, BMP, ...\)](#) or even as complete "3D scene" saved in [3D scene file \(VRML, IV, OOGL, ...\)](#).

Please proceed with following steps:

1. Create a [3D display](#) in AnTherm and customize its representation according to your needs.
2. Open the application commonly used by you to create reports (e.g. MS-Word) optionally applying your personal templates, footnotes, headings a.s.o..
3. Copy the Text from the [project description](#) to the clipboard (right mouse button, copy) and then transfer it into your report (right mouse button, paste).
4. Copy the [graphical 3D display](#) to the clipboard (use the [button Copy image](#)) and transfer it into your report (right mouse button, paste).

Where to find the calculated f_{Rsi}^* -values?

Temperature factors f_{Rsi}^* are output in [Results report](#). The calculation f_{Rsi}^* values assumes, that the coldest space represents exterior space. AnTherm searches and calculates coldest points on surfaces for all other spaces. This points are output in the [Results report](#) together with its coordinates, surface temperatures and corresponding f_{Rsi}^* values.

See also: [Temperature factor fRsi](#)

According to EN ISO 10077-2, when calculating the thermal conductance coefficient of a window profile, one needs two dimensional thermal conductance (Leitwert) L^{2D} in W/mK. AnTherm shows conductance matrix in W/K. Which length shall be applied in the division to get correct L^{2D} value?

For a [2D project](#) Antherm will report the thermal conductance values directly. AnTherm calculates two dimensional construction as homogenous in third dimension and having an extent of 1 meter.

While within a [3D project](#) you can calculate L^{2D} only if you take care of keeping the construction homogenous in one of the three dimensions. Then divide by the thickness of the model in this direction - however we recommend to use [2D project](#) to calculate L^{2D} only.

In between the messages shown during the calculation (in [Solver window](#)) there is a message telling "error", but the computation continues and results are shown. Is there any error in construction entry? Can I trust the results?

Under certain, rare circumstances, the solver will fire few "error/warning" messages. If the condition which led to such message has no influence onto calculation results the program will step further and continue with the computation. To keep things simple there are design measures taken, taking care to provide computational results if the construction input has no formal errors. If this is not the case the program will stop the calculation, show some diagnostic message and will not create any results at all.

According to this workflow within calculation step one has to analyse such [warning or error message from](#)

[calculation step](#) only if the program has not shown any results..

How can I create a two dimensional model (2D project)?

The "[type of the project](#)" is set up during creation of the project itself; choose: [. File→New→2D/3D-Schichten/3D](#). A [2D project](#) can always be [converted](#) to [3D layered project](#). Upon your need one can convert a [3D layered project](#) to [3D project](#). Converting of a three dimensional project to a [2D project](#) is not always possible!

During evaluation of 2D projects it is advised to reduce the analysis solely to [Z slice plane](#) - having turned perspective view off also.

Does two dimension simulation result in correct results if it has been input in tree dimensions (i.e. as 3D project)?

All results are correct and can be immediately used for further calculations if the thickness of the layer (in a layered 3D project) is kept at 1000 mm, or, for a 3D project, all elements share the same Z dimension of 1000 mm.

Please note, that in such case all elements of the [conductance matrix](#) have exactly same values as those of the length related conductance matrix. Only the unit of W/K has to be replaced with W/(m*K).

Is it possible to view results on slices through the construction (to cut through)?

Yes. Slicing (or cutting) through evaluation results is possible on [slices](#) parallel to axes planes - with the only exception to dew point values (those are defined on the construction's surface only).

Follow following steps to get visualizations of sliced results:

1. Within [Results 3D](#) view select the view direction you wish (either within [3D panel](#) or from the [view menu](#)).
2. Turn off the visibility of the [Surface](#) (or make it transparent) to see the interior of your model.
2. Turn (optionally) the perspective off (in the [view menu](#) or in the [panel General](#)).
3. Select one of the [slice plane panels \(x, y or z\)](#) and set the slice [Position](#) of your choice.
4. Select the [function](#) which values shall be visualized.
5. Now you can combine the display with [isolines](#) by turning [them on](#) and optionally [making the slice surface transparent](#).
6. You can combine the view further if you want. For example turn the [model](#) on.

How to get the detailed view of the model construction?

By activating the [model](#) edges line view within the 3D-View material boundaries can be shown on surface and/or slices.

The solid view of the [model](#) can be shown as exploded parts by applying a [shrink](#) factor.

How to transfer or combine, import, paste or insert an already entered constructions (e.g. a complex Isokorb) from other projects into the one currently processed?

Preferred method of combining projects is by using the clipboard function of [copy](#) and [paste](#). You will find them in [the context menu of element editing](#). You can follow exemplary steps below:

- Save the project which shall receive the other construction inserted.
- Open the project containing the other component you want to reuse.
- Select those (or all) elements you want to reuse in the other project (e.g. use the [select function of the context menu](#)).
- [Copy](#) selected elements (the data of them) onto the clipboard.
- Reopen the target project (the one you want to insert elements to).
- Select the element after which the elements from the clipboard shall be inserted.
- [Paste](#) elements of the construction on the clipboard into current project. All inserted elements will be automatically selected for further processing (for group assignment or position adjustments).

- Use editing functions to [translate](#), [rotate](#) etc. to position the inserted component elements into its final place within your construction.
- Save the project.

How to create a 3D project from a DXF file (import)?

Currently the [import function from DXF file](#) interprets such files as two dimensional drawings. By using [element grouping](#) latched on one of DXF attributes when importing the file it is possible interpret such element groups as layers of [3D-layered project](#) and to process them accordingly.

Use following process steps:

1. Use one attribute to assign DXF objects to groups of 3D-layers (independently of the assignment of other attribute for the purpose of material identification).
2. In the DXF import settings select that attribute for grouping
3. The imported project (import always creates a [2D project](#)) shall now be [converted to a 3D layered project](#). After such import all elements will be located on the first layer.
4. Create a [new layer](#).
5. Go to the first layer and [select elements by group](#). Group names are taken based on the grouping [import setting](#) set during the import.
6. By using the [context menu function "Cut"](#) place all selected elements on to the clipboard. This function will also remove the elements from the model, that these cut element are now only on the clipboard.
7. Go to the new layer created above an by using the [context menu function "Insert"](#) paste elements from the clipboard to that new layer.
8. Repeat steps 4. through 7. for all further groups until elements of each group have been moved to distinctive layers.
9. Assign [thickness to all layers](#) properly.
10. Save the project.

How to calculate Psi-values with AnTherm?

The [Psi-Value Determination](#) provides automatic calculation of the Psi-Value for a two dimensional (linear) thermal bridge. The calculation is based on the [conductance matrix](#) output by AnTherm and the U-Values output in [Modelling Report](#) for layered construct at adiabatic cut-off planes determined automatically.

The additional [tool "Psi-Value Calculator"](#) allows quick calculation of linear correction factors for any arbitrary thermal bridge also.

It shall also be considered, that the calculation of Psi-Value depends on the used one dimensional model and it is ambiguous. Furthermore, reasonable definition of Psi-Value applies to two space cases.

Will future version of AnTherm calculate and output Chi-values also?

Currently one can calculate Chi-Values by simple manual calculation based on the [conductance matrix](#) output by AnTherm automatically. However it is planned to provide a very simple tool for calculating Chi-Values automatically in the future.

It shall also be considered, that the calculation of Chi-Value depends on the used one dimensional model and it is ambiguous. Furthermore, reasonable definition of Chi-Value applies to two space cases.

The licensing procedure of this application requires a MAC address of physical network LAN adapter. How to discover the MAC address?

Complete listing of all adapters available for licensing shall be found in the [About window](#), output just beyond the licensing information. To otherwise retrieve MAC addresses of physical LAN adapter ne can use the tools of the operating system (e.g. by using a command "ipconfig /all"). Alternatively, there is a tool application installed along with AnTherm to its installation folder called „*adaptersAddresses.exe*“ which will show adapters available in your system suitable for license activation.

Remark: As an alternative of binding the license to the computer by its network adapter (i.e. via



the MAC address) there is an [activation option with a hardware dongle](#) which adds mobility of the license between multiple computers.

See also: [License activation procedure](#).

An exception message "SAFcdtsti crash" or "security exception SHA1" is shown.

This exception is catch due to severe damage of the installation. It indicates possible damage to one or more binaries of the application components. You are notified not to use program's results for any purpose - under no circumstances. Caused by such severe error results could be in error too!

Please report such conditions to the vendor. Analysis and sufficient measures shall be started immediately. Thank you!

A message "CrystalReports.Engine.LoadSaveReportException" is shown instead of all reports.

This exception is catch due to severe damage of the installation of the [reports engine](#) used. It indicates possible limitation of user rights to access application components. Typically this occurs if the software has been installed for exclusive use by one user and accessed from an account of some another user.

To resolve the issue you are advised to install the software for all users instead for the one user only. Uninstalling and installing the software by the user with administrative right will be required.

Advice: If the software shall be run by different user than the one who installed it, then the **installation for all users** must be chosen (the choice has to be done within the dialog while selecting the installation folder)!

[Reinstalling the Crystal Reports Runtime Components manually](#) shall resolve the issue.

On a computer without the drive C: a message "CrystalDecisions.Shared.CrystalReportsException: The report could not be loaded. ---> System.Runtime.InteropServices.COMException (0x80004005): The system cannot find the path specified" is shown instead of all reports.

Manual [reinstallation of Crystal Reports Runtime Components](#) does not resolve the issue.

This results from erroneous behaviour of Crystal Reports outside of our influence. Crystal Reports Runtime assumes the existence of drive C:. If that drive is not existent the above message will be shown - the Crystal Reports installer application sets registry values that point to the C drive.

WORKAROUND 1: Create the drive C: (eg. set some USB Drive to C: within Disk Management).

WORKAROUND 2: Use the following steps to change respective registry values in the Registry Editor.

=====

WARNING: The following resolution involves editing the registry. Using the Registry Editor incorrectly can cause serious problems that may require you to reinstall the Microsoft Windows operating system. Use the Registry Editor at your own risk.

HELP: For information on how to edit the registry key, view the 'Changing Keys And Values' online Help topic in the Registry Editor (Regedit.exe).

RECOMMENDATION: It is strongly recommended that you make a backup copy of the registry files (System.dat and User.dat on Win9x computers) before you edit the registry.

=====

1. On the 'Start' menu, click 'Run'.
2. In the 'Run' dialog box, type "Regedit" then click 'OK'.
3. In the Registry Editor browse to the following subkey:
HKEY_LOCAL_MACHINE\SOFTWARE\Business Objects\10.5\Report Application Server\InProcServer\LocalConnectionMgr
4. Right-click the 'ConnectionDirectoryPath' string value, then click 'Modify'.
5. In the 'Value data' text box, change "c:\" to the drive letter where the following directory exists (You may need to search your computer to find where this directory exists, i.e \Program Files\Common Files\Business Objects\2.8\bin) Click 'OK'.

If above does not fix the problem:

6. Right-click the 'LocalConnectionMgr' subkey, click 'New' then click 'String Value'.
7. Name this String Value "ReportDirectoryPath".
8. Right-click the 'ReportDirectoryPath' string value and click 'Modify'.
- 9 In the 'Value data' text box, type the same drive letter as you typed in step 5.

=====

NOTE:After making changes to the registry, restart the application as required.

3D widows are empty or black

or windows shown on top of 3D windows flash rapidly

or 3D visualization is corrupt, incomplete or 3D-surfaces look distorted.

An exception message "SEHException" in vtkWin32RenderWindowInteractor, InitializeVtk, vtkControl.OnCreateControl is shown.

This exception is very rare, and there are some known conditions which can be worked around:

On some graphics systems (graphics adapters, cards) the initialisation of 3D windows cannot be completed successfully if the setting of graphics acceleration of the operating system is active.

We have seen such a conflict between 3D software components and the setting of the 2D acceleration on only few systems running Windows XP – symptoms shown where empty (black) 3D windows in AnTherm or even surprising crashes with no chance of reproducing the conditions. By turning the 2D acceleration of on these system all such symptoms disappeared then.

Instruction: Go to the Windows control panel, then go to the setting of your graphics adapter (Display->Tab Settings->Advanced->Troubleshoot). The "acceleration" is set to highest by default. Try reducing the setting, or even turn it off. Then reboot the system prior to testing the condition with AnTherm again.

Please report such and alike difficulties to the vendor too. Thank you!

What shall be done if the program does not respond at all?

At first please verify if you're not waiting for computation results to complete. There is either a message shown in the [Solver window](#) to inform you about the progress. Te computation can be stopped by pressing the Stop button. It the application is not responding for a longer period, please use the operating system's task manager to finish AnTherm as the last resort. The task manager can be reached by simultaneously pressing the keys Ctrl-Alt-Del. Shall such scenario occur repeatedly, then please do not hesitate to report this behaviour to us immediately. Please attach the specific AnTherm project file to allow further analysis. Thank you!

How to backup complete simulation results of one project?

How to find the [folder of simulation results](#) ?

Intermediate results (simulation results) of each computation belonging to one project are stored to [subfolder](#) "[<projectfile>.dir](#)".

To avoid accidental manipulation of these results AnTherm applies the hidden attribute to each such simulation results folder. By default such hidden objects are not shown within Windows file explorer; they are kept hidden from the user. Hidden file system objects are only shown if the setting "show hidden files" is tuned on in windows file explorer, thus also showing folders of simulation results created with AnTherm.

The setting described above can be set in *Windows Explorer* by selecting the *Menu Tools->Folder Options...* from the tab *View* . Change the setting for *Hidden files and folders*.

AnTherm creates hidden data (directories) in significant amounts. How does this data relate to AnTherm projects?

How to remove simulation results folders when they are not needed anymore?

Intermediate results (simulation results) of each computation belonging to one project are stored to [subfolder](#) "[<projectfile>.dir](#)". To avoid accidental manipulation of these results AnTherm applies the hidden attribute to each such simulation results folder.

One can remove this data at any time because AnTherm can recreate this data (recalculate) from project's information if needed.

To ease the consistent removal process there is a [Tool "Simulation Folders Cleanup"](#) available to the user.

On my notebook only one side of slice planes is coloured, the other side is displayed in black (notebook with Intel HD Graphics, Windows XP).

This is related to the OpenGL implementation of the graphics card and combination of system settings - the FrontFace-Property is not automatically propagated to BackFace-Property (which is not set and by that defaults to FrontFace).

WORKAROUND: Within Application Settings please turn on the [Extended OpenGL Setting "ExplicitBackfaceProperty"](#) (slight performance loss can be eventually observed).

After overwriting the activation file AnTherm.HID in the installation folder AnTherm still uses the earlier version of the file.

Replacing the activation file AnTherm.HID within installation folder has no effect to AnTherm.

The preferred method of activation is by placing the file AnTherm.HID into any user folder and setting the path to that file within the [application setting "alternative license file AnTherm.HID"](#). On computers running Vista overwriting the activation file with new content is protected by the file virtualization enforced on system folders automatically. Therefore the activation by copying the file AnTherm.HID into the installation folder is effective only once on these systems.

See also: [License activation procedure.](#)

See also: [Short reference](#)