

VOLTRA

computer program to calculate
3D and 2D transient heat transfer
in rectangular objects

version 9.0

© 2026 PHYSIBEL

physibel

This manual is intended as a complete reference for the VOLTRA application. Documented software application examples, tutorials & exercises, short practical guides (How To's) and validation cases are accessible via the Physibel Knowledge base (www.physibel.be/en/knowledge).

Table of contents

CHAPTER A INTRODUCTION. 4
A.1 Program overview 4
A.2 VOLTRA brief history 5
A.3 Technical specifications 5
A.4 Licence 5
A.4.1 Hardware licence 6
A.4.2 Software licence 6
A.5 User Account Control (UAC) 6
A.6 Overview of delivered files and file structure 7
A.7 Logging 7
A.8 Text convention 7
A.9 Program settings 7
A.10 General editing commands 9
A.11 Program parameters 9
CHAPTER B EDITING INPUT DATA. 10
B.1 VOLTRA start up 10
B.2 File commands 10
B.3 VOLTRA program layout and input windows . . . 12
B.4 Document notes 14
B.5 Grid window 14
B.6 Grid functions 15
B.6.1 Insert mesh 15
B.6.2 Grid mesh selections 15
B.6.3 Grid unit 16
B.6.4 Create minimum grid 16
B.6.5 Split (selected) mesh(es) 17
B.6.6 Auto split 18
B.6.7 Merge selected meshes 18
B.7 Minimum Grid window 18
B.8 Blocks window 19
B.9 Block functions 20
B.9.1 Insert block 20
B.9.2 Block selections 21
B.9.3 Delete, copy, move 22
B.9.4 Transform; insert file 23
B.9.5 Revolve 23
B.9.6 Arrange blocks 24
B.9.7 Reduce blocks to clipped object and colours 24
B.9.8 Detect void (adiabatic) spaces within object 25
B.9.9 Redefine colour globally 25
B.10 Colours window 25
B.10.1 Colour number 26
B.10.2 Colour type 26
B.10.3 Standard (last column) 27
B.10.4 Colour subtype 27
B.10.5 Colour physical heat flow direction 28
B.10.6 Colour geometric heat flow direction 28
B.10.7 Colour name 29
B.10.8 Physical properties 29
B.11 Colour functions 30
B.11.1 List colours 30
B.11.2 Load material properties 31
B.11.3 Split zones 32
B.11.4 Copy - paste 32
B.12 Functions window 33
B.12.1 Internal time functions (types STEP0, STEP1, STEP2, SINE, and SINPOS) 34
B.12.2 External functions (type FILE) 36
B.12.3 Weather data files (types EPW and TMY3) . 37
B.13 Function Graph window 38
B.14 Function commands 39
B.15 Node BCs window 39
B.16 Borderface BCs window 40
B.17 Material BCs window 40
B.18 Solar data 41
B.18.1 Real sun path 42
B.18.2 Fixed sun position 43
B.19 Sun obstacles window 43
B.20 Controls window 45
B.21 Output nodes 47
B.22 Output blocks 48
B.23 Ventilation flows window 49
B.24 Report Definition window 49
B.25 Report Frequency 52
B.26 Save Frequency 53
CHAPTER C VIEWING THE MODEL. 54
C.1 Image submenu 54
C.2 Viewpoint 54
C.3 Zoom commands 55
C.4 Projection 55
C.5 Object appearance 56
C.6 Quick block selection using the mouse 58
C.7 Active item feedback in the Image window 58
C.8 Show North 58
C.9 Pick temperature using the mouse 59
C.10 Object clipping 59
C.11 Colour clipping 60
C.12 Coordinates window 60
CHAPTER D CALCULATION. 62
D.1 Automatic calculation of thermal properties . . . 62
D.1.1 Colour type EQUIMAT 62
D.1.2 Colour type BC_SIMPL or BC_FRE_S 64
D.1.3 Colour type BC_FREE 65
D.1.4 Colour type BC_SKY or BC_NOSKY 66
D.1.5 Colour type TRANSMAT 66
D.2 Calculation parameters 67
D.2.1 Time 67
D.2.2 Iterations 68
D.2.3 Radiation 69
D.2.4 Automatic calculation of thermal properties 70
D.3 Calculation of view factors 70
D.4 Calculation of solar fluxes 71
D.5 System calculation 72
D.6 Calculation principles 73
D.7 Unlinking air temperatures in cavities of type BC_FRE_S or BC_FREE 74
D.8 Processing ventilation flows 74
D.9 Solar radiation processing 75
D.9.1 Definition of solar properties ρs and τs for MATERIAL 76
D.9.2 Irradiance: solar load to external surfaces . . 77
D.9.3 Absorption of solar radiation 79
D.9.4 Processing of reflection 79
D.9.5 Transmission to internal zone: solar load to internal surfaces 80
D.9.6 Solar load to opaque surfaces behind transparent materials 81

D.10 Heat flow divergences 82
D.11 Batch calculation 83
D.12 Command line program execution 84
CHAPTER E GRAPHIC OUTPUT..... 86
E.1 Graphic Output window 86
E.2 Animate functions 86
E.3 Save animation images 87
E.4 Projections and clip object 87
E.5 Zoom commands 88
E.6 View submenu 89
E.6.1 Lines 89
E.6.2 Fill modes 89
E.6.3 Sun position and obstacles 90
E.6.4 Sun shaded image 91
E.6.5 Temperature profile along a grid line 93
E.7 Settings 94
CHAPTER F TEXT OUTPUT..... 98
F.1 Text Output window 98
F.2 File commands 98
F.3 View options 99
F.3.1 Input data 99
F.3.2 Results per time step 100
F.3.3 Report output 102
F.4 Settings 103
F.4.1 Output precision 103
CHAPTER G VOLTRA TEMPLATE FILES..... 104
G.1 General 104
G.2 Colour database 104
CHAPTER H PROGRAM FUNCEDIT..... 105
H.1 Introduction 105
H.2 Function type 106
H.3 Function parameters 107
H.4 Function data 108
H.5 Graphic parameters 108
H.6 Using a spreadsheet program to create functions 110
CHAPTER I RADCON MODULE..... 112
I.1 Purpose 112
I.2 Implementation 112
I.3 Table for black radiation heat transfer coefficient 116
I.4 Table for emissivities 117
I.5 Tables for convective heat transfer coefficient .. 118

CHAPTER A INTRODUCTION

A.1 Program overview

VOLTRA is a thermal analysis program for transient heat transfer in three-dimensional rectangular objects. VOLTRA is an extension for time-dependent boundary conditions of TRISCO (for steady-state heat transfer).

As in TRISCO, the geometry is described with a list of rectangular blocks, the vertices of which lie on grid points of a rectangular grid. Materials and surface boundary conditions with different thermal properties are identified using separate colours. Each geometry block is part of either a material or a surface boundary condition region, and has a reference to one of these colours. Node boundary conditions with fixed or time-dependent temperature or power are possible and can be placed in grid point locations. Also border face boundary conditions in the interface between two colour regions with fixed ort time-dependent temperature or heat flux, or material boundary conditions with fixed or time-dependent temperature or heat power density are possible.

A TRISCO data file, Trisco2D data file, BISTRA data file, BISCO data file or bitmap (.bmp file) can be taken as starting data in VOLTRA.

The time-dependent boundary conditions are described with functions, either built-in functions based on parameters, external user-defined functions (saved as separate files) based on function values given at a fixed time interval, or standardised weather files.

The thermal conductivity, specific heat and reflection factor of a material can refer to functions described in external text files: thermal conductivity as a function of temperature, specific heat as a function of temperature, solar reflection factor as a function of angle of incidence.

Dynamic solar heat gains can be studied. To this end a solar processor is built in (which requires the use of the RADCON module, see Chapter I). An object image with shadows cast by direct sun light at any time and any geographic location can be viewed. The direct and diffuse solar radiation from climate data (e.g. as given in the European Test Reference Years) is cast on the material surfaces. The absorbed solar radiation is converted to time-dependent node powers, as additional boundary conditions to the system. Transmission of solar radiation is possible via transparent materials to simulate heat gains through windows. The reflected solar radiation is diffuse and distributed according to the view factors (calculated with the RADCON module) to other material surfaces. Direct radiation is reflected (as diffuse radiation) using a reflection factor which may be function of the angle of incidence.

VOLTRA can be used to carry out fire simulations (with use of the RADCON module, see Chapter I). Radiative heat transfer becomes more important when temperature differences increase. The RADCON module allows to include the simulation of non-linear radiation based on geometric view factors. The thermal conductivity and specific heat of materials may be temperature dependent. Latent heat is modelled through an increase of specific heat within a small temperature interval around the change of phase temperature.

Heat transfer to objects via (time-dependent) ventilation between zones at different temperatures can be studied.

Controls are available to replace boundary condition functions or material property functions with other functions, based on sensor temperature target functions.

VOLTRA allows creating time-dependent graphic animations of moving shadow patterns, calculated temperature field and heat flux field in the studied object. Alphanumeric lists of time functions of temperatures in individual nodes or heat flows through given surfaces through the object can be made.

The RADCON module (see Chapter I), which allows a better simulation of radiation (using view factors) and convection, is available in VOLTRA, and is required for fire simulation and use of the solar processor.

A.2 VOLTRA brief history

VOLTRA was first developed in 1993 based on the steady-state program TRISCO.

- Version 1.0 was ready in May 1993.
- Version 3.0 (December 2001) was the first Windows version.
- Version 4.0 (April 2003) included temperature dependent material properties and simulation of ventilation flows through different zones.
- Version 6.0 (September 2006) included the solar processor based on direct and diffuse solar radiation weather data files.
- Controls were introduced in version 8.0 (May 2017).

The detailed version history can be found on the website: www.physibel.be/en/products/voltra/versions .

A.3 Technical specifications

- VOLTRA is a 64-bit Windows program. VOLTRA is installed in C:\Program Files (x86)\Physibel\VOLTRA9.
  In this folder the executable file is VOLTRA.exe.
- The standard version of VOLTRA allows a maximum of 5,000,000 nodes. An add-on feature to calculate an unlimited amount of nodes is available for purchase (feature Unlimited Nodes). The maximum number of nodes then depends on the RAM installed.
- VOLTRA relies on OpenGL for use of hardware accelerated graphics.
- VOLTRA is protected by either a hardware key, provided by Physibel, or a software licence (A.4).

The Physibel 64-bit programs need several shared libraries (.dll files) from Microsoft.

The Microsoft Visual C++ 2017 Redistributable Package (x64) is included in the installation files of VOLTRA, but can also be downloaded from https://support.microsoft.com/en-us/help/2977003/the-latest-supported-visual-c-downloads.

This package installs mfc140.dll and other files in the directory C:\Windows\SysWOW64, required by 64-bit applications developed with Microsoft Visual C++ 2017.

A.4 Licence

The Physibel applications are protected by a licence, to prevent illegal use of the software. This can be either a hardware licence (Sentinel USB key) or a software licence.

A.4.1 Hardware licence

Perpetual licences are protected by a hardware licence (Sentinel USB key). The licence is locked to this dongle. It is strongly recommended to take a financial assurance for loss of the key by theft or any other incidence.

Before VOLTRA can be used, the driver (delivered with purchase) for the hardware key needs to be installed. Administrator rights are needed for this installation. It might be necessary to restart your computer after installing the driver.

During use of VOLTRA, the dongle needs to be attached to the computer at all times.

A.4.2 Software licence

Subscription licences are protected by a software licence. After first installation of VOLTRA, the user needs to activate the licence using the activation code received upon purchase of the software. When first opening VOLTRA, the Software Licence Manager will open automatically and prompt the user to activate the licence, or to register in the network floating database, depending on the licence type. The computer needs to be connected to the internet for this licence activation. Cloud-based floating licences need to be activated every time when opening the application.

After activation, the Software Licence Manager is available from the menu for stand-alone licences:

Help → Software Licence Manager...

This allows the user to check the expiry date of the licence subscription and be redirected to renew the licence.

Users with a floating network licence can check out a licence from the database for a maximum period of 3 days.

Help → Check out/in floating licence...

The software licence key for VOLTRA is linked to the computer where it was activated. When you want to move the application with a stand-alone licence to another computer, you can deactivate the licence in the Software Licence Manager. The licence can then be reactivated on a new computer.

If the Software Licence Manager is unavailable from the application itself, you can also open it from the command prompt. You need to first change directory to the Program Files folder where the application is installed. Then, open the Software Licence Manager with the following command:

QlmLicenseWizard.exe /settings "VOLTRA 9.0.lw.xml"

A.5 User Account Control (UAC)

Administrator rights are necessary to install VOLTRA on a computer.

The installation of the application is machine-level, meaning all user accounts can access the application after installation.

Each user account has its own copy of the colour database (see G.2), to work with. These files can be found in the user's application data directory (C:/Users/*username*/Appdata/Roaming/Physibel/VOLTRA9/).

In case of a software licence, the licence for VOLTRA is linked to the computer, but is stored at the user level. Each user account on a computer has to activate the licence in order to have access to the licence.

A.6 Overview of delivered files and file structure

VOLTRA is installed (when the user has accepted the default destination) in the folder C:\Program Files (x86)\Physibel.

The following files are installed

Folder	*File*	*Description*
...\Physibel\VOLTRA	VOLTRA.exe	VOLTRA (64-bit) program
	*.dll, *.pat, *.rgb	Program resources
	QlmLicenseWizard	Software Licence Manager files
...\Physibel\Materials	*.phm	Material databases
...\Physibel\Manuals	VOLTRAManual.pdf	VOLTRA manual

The colour database file ColourDatabase.vtr (see G.2), together with the parameter file VOLTRA.prm (see A.10), are installed in C:/ProgramData/Physibel/VOLTRA9 (machine-level), and copied to each user account's application data folder when first opening VOLTRA (C:/Users/*username*/AppData/Roaming/Physibel/VOLTRA9).

A.7 Logging

When using VOLTRA, information is continuously written to a log file located in the user account's application data folder (C:/Users/*username*/AppData/Roaming/Physibel/VOLTRA9). This file contains information that can help for debugging when something went wrong, and also to clarify or elaborate on error or warning messages shown in the application.

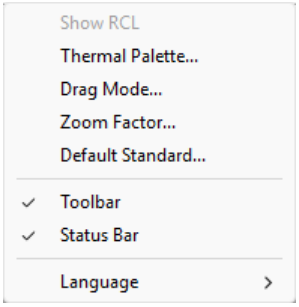
A.8 Text convention

Menu paths are printed in *italic*, with format *Submenu* → *Command*.

E.g. *File* → *Open* (Open command in File submenu).

A.9 Program settings

Global program settings are available via the Settings submenu.



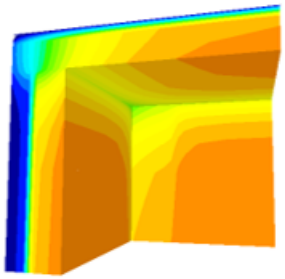
- *Settings* → *Show RCL*: to change the coordinate system from XYZ (right-handed Cartesian coordinate system, using axes X, Y en Z) to RCL (coordinate system using rows, columns and layers – as defined in the initial version of VOLTRA).

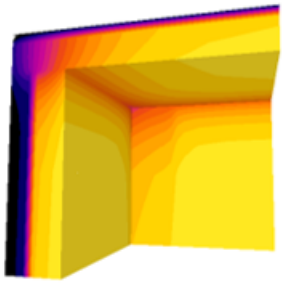
The XYZ system is the default system for new VOLTRA data (from *File → New*).
Settings → Show XYZ: to change the coordinate system from RCL to XYZ.

- *Settings → Thermal Palette...*: to select the thermal palette used to visualize the temperature gradient in the solution.

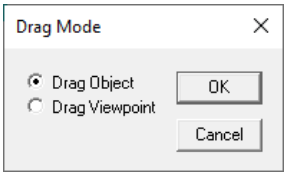


The user can choose between the traditional Physibel 'Rainbow' thermal colour palette, or the 'Ironbow' thermal colour palette, which is often used by IR camera's.



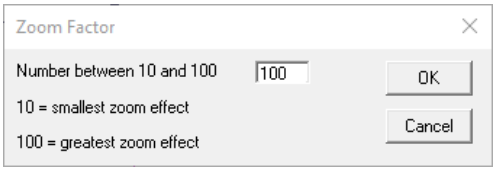


- *Settings → Drag Mode...:* to define how the view of an object (in the **Image window** or the **Graphic Output window**) changes as a result of a rotate or pan command (using the mouse or an arrow key).

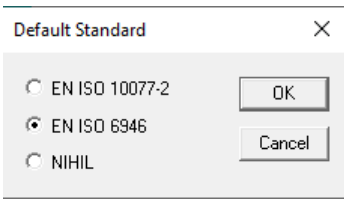


"Drag object" moves the viewpoint position with respect to the object. "Drag viewpoint" moves the object with respect to the viewpoint.
The selected drag mode is saved in the program parameter file (see A.10).

- *Settings → Zoom Factor...*: to customise the zooming factor of the mouse scroll wheel. The selected zoom factor is saved in the program parameter file (see A.10).



- *Settings → Default Standard...*: set EN ISO 10077-2, EN ISO 6946 or NIHIL as default standard for the definition of colour types.



- *Settings → Toolbar*: presence of the toolbar giving access to menu commands through icons.
- *Settings → Status Bar*: presence of the status bar at the bottom of the VOLTRA application window, containing context depending information.
- *Settings → Language*: the selected language (English, Dutch, French, German) becomes effective the next time the program is started.

A.10 General editing commands

Copy – paste

- *Edit → Copy Value*: copies the value from the active input cell into the clipboard.
- *Edit → Paste Value*: copies the value from the clipboard into the active input cell.

Undo – redo

- *Edit → Undo*
  Undoes the last edit action (e.g. to cancel a mistake).
  This command can be repeated to undo more previous edit actions.
  Only the last 100 edit actions are stored for undo.
  The undo command is impossible after saving a file, closing the current file or starting a calculation.
- *Edit → Redo*
  Restores the previous edit action that was made undone using *Edit → Undo*.
  In other words: "redo" undoes "undo".
  This command can be repeated to restore more edit actions that were made undone.

A.11 Program parameters

The global default values (calculation parameters, output parameters, interface language, graphic report definition, grid unit, etc.) are saved in the file VOLTRA.prm in the application data folder (C:/Users/*username*/Appdata/Roaming/Physibel/VOLTRA9). When the program VOLTRA is started, the program parameters are loaded from the parameter file.

The program parameter default values can be changed from several input dialog boxes. When the button "Set As Default" in an input dialog box is clicked, the corresponding program parameters are saved in the program parameter file VOLTRA.prm at exiting VOLTRA.

CHAPTER B EDITING INPUT DATA

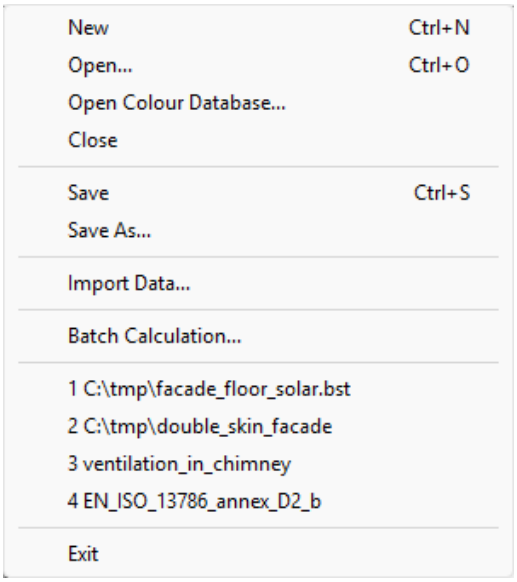
B.1 VOLTRA start up

When VOLTRA is launched, the parameter file VOLTRA.prm is read (see A.11) to load global program settings.

Then the colour database ColourDatabase.vtr is read (see G.2) to initialise material and surface boundary condition properties of the **Colours window**.

There is no visual feedback in the initial screen of VOLTRA to indicate if VOLTRA.prm and/or ColourDatabase.vtr are not found and thus not read.

B.2 File commands



File → New...

Opens a new project, after closing the current project (if present). The new project starts off with a single cubic block with sides equal to the grid unit (see B.6.3).

All parameters are reset to their default values (as stored in the parameter file VOLTRA.prm), except for the colour definitions. This allows to reuse the colour definitions of the previous open file.

File → Open...

Opens an existing .vtr, .trc or .tr2 file by selecting it in the File Open dialog box. When a file is opened, all parameters are reset to their default values (as stored in the parameter file VOLTRA.prm), except for the colour definitions. This allows to reuse the colour definitions of the previous open file (cf. *File → New*). Then all data from the chosen VOLTRA data file are read and loaded into memory. Colour definitions that are not present in the data file, remain untouched.

File → Open Colour Database...

The colour database contains predefined material and boundary condition properties (stored in the file ColourDatabase.vtr in the application data folder, see G.2), which is loaded

at every VOLTRA start-up (see B.1). It is advisable to add frequently used materials and boundary conditions to the colour database.

These predefined colours of the colour database are then available for new colours in any document.

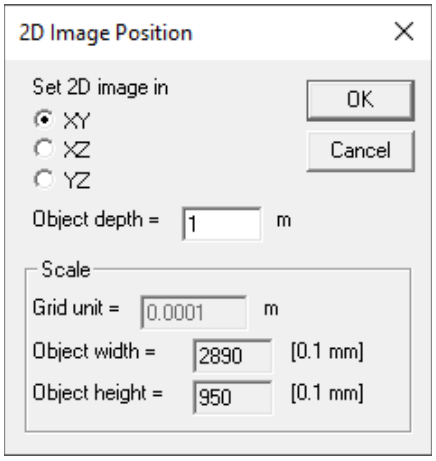
File → Import Data...

This command allows to create a new VOLTRA data file starting from a TRISCO data file, a Trisco2D data file, a BISTRA data file, a BISCO data file, or a bitmap.

Choose the wanted file type in the File Open dialog box.

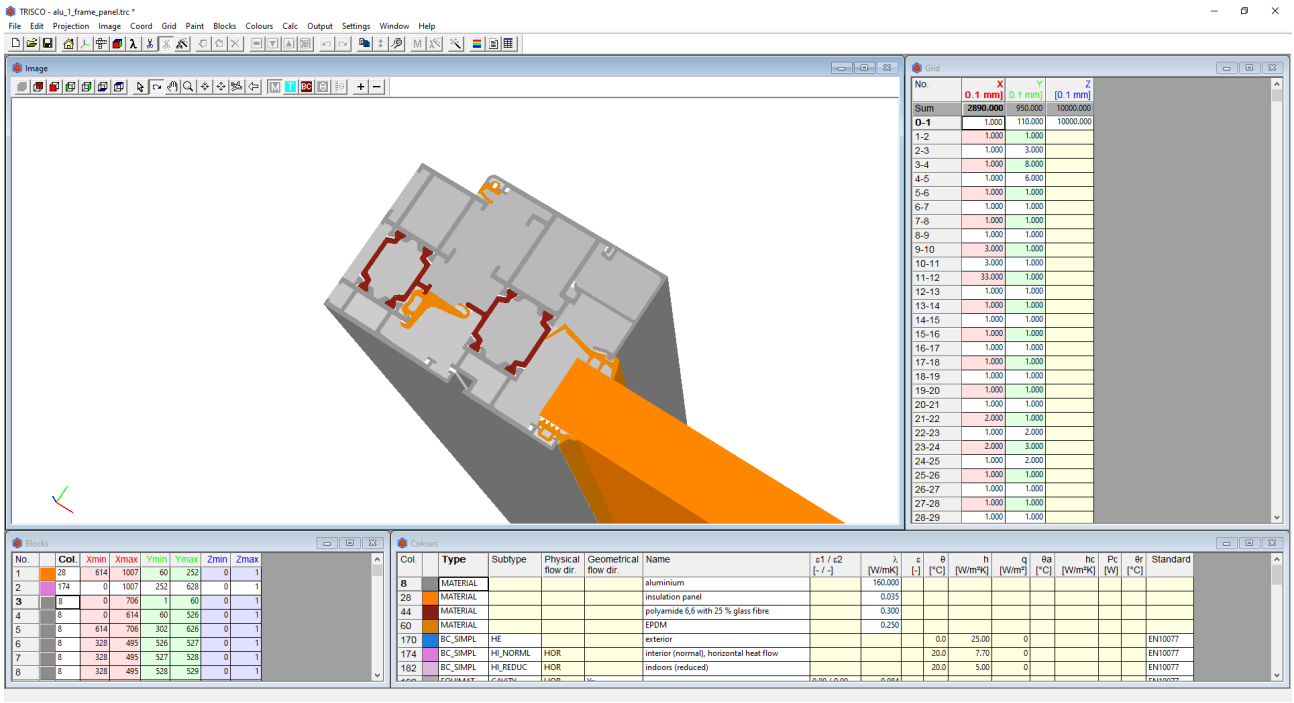
.bsc files and .bst files contain a link to a bitmap file (256 colours .bmp file). The bitmap is extruded (to get a third dimension) and converted into a list of blocks defined in a minimum grid (i.e. having grid planes only at pixel changes in the bitmap). The grid unit corresponds to the pixel size of the bitmap, as determined in BISCO or BISTRA. The bitmap can be set in XY, XZ or YZ. The third dimension is made of 1 grid mesh, the size of which is a user input (Object depth).

The grid unit, object width and height are read from the BISCO or BISTRA data file and shown for information.



The new blocks can use either superposition (with possible overlaps of blocks, which normally minimises the number of blocks) or juxtaposition (without any overlap of blocks), dependent on the user's answer to the prompt "Use superposition of blocks?".

The thermal properties of the colours in the bitmap are copied from the data in the .bsc or .bst file.



When a BISTRA data file (with extension .bst) is imported, the report definition is not read and must thus be redefined in VOLTRA (see B.24). The north orientation (see B.18.1) or fixed sun position (see B.18.2) is read, though in the former case the angles defined in BISTRA cannot always be converted to the angle definitions in VOLTRA. In this case a warning message is shown prompting the user to redefine the north orientation.

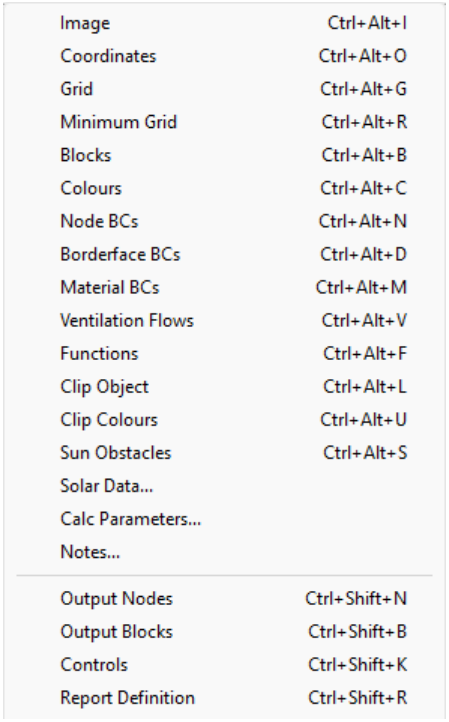
B.3 VOLTRA program layout and input windows

The input in VOLTRA is organised in several windows. When a VOLTRA document is opened, the following windows are opened:

- **Image window** (top left) with a graphical representation of the model. In this window the object can be interactively rotated and zoomed in.
- The **Grid window**, with the distances between the successive grid planes in three dimensions.
- The **Blocks window**, with a list of all building blocks of the geometry.
  All coordinates refer to the grid numbers in the **Grid window**.
- The **Colours window**, with the thermal data of all materials and surface boundary conditions. Each block in the **Blocks window** refers to a colour in the **Colours window**.

A window can be activated by clicking with the left mouse button on its title bar or in its client area (when the window is already open and visible on the screen), or the window can be opened and activated by selecting the corresponding command in the Edit submenu.

Upper part of Edit submenu

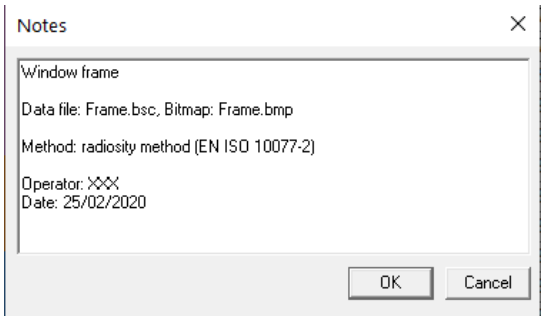


Other input windows are:

- The **Coordinates window**, to view the absolute coordinates of the grid planes (which are orthogonal to the coordinate axes).
- The **Minimum Grid window**, for the definition of minimum grid mesh widths.
- The **Node BCs window**, for the definition of node boundary conditions.
- The **Borderface BCs window**, for the definition of boundary conditions in border faces between two colours (fixed or time-dependent temperature or heat flux).
- The **Material BCs window**, for the definition of boundary conditions inside material colours (fixed or time-dependent temperature or volumetric heat power density).
- **Ventilation Flows window** with ventilation flow paths.
- **Functions window** with functions.
- The **Clip Object window**, for the definition of cut-off planes to get a clipped detail (of the total object) in the **Image window**.
- The **Clip Colours window**, for the definition of colours to cut away in the **Image window**.
- The **Sun Obstacles window** with definition of obstacles around the model.
- The **Report Definition window** with selection of reported outputs.
- The **Output Nodes window** with coordinates of the nodes for which a time-dependent output is required, or which are used as sensor points in controls.
- The **Output Blocks window** with coordinates of the blocks for which the time-dependent conduction heat flowing through its faces is to be reported.
- The **Controls window**, for the definition of a temperature control (heating or cooling).

B.4 Document notes

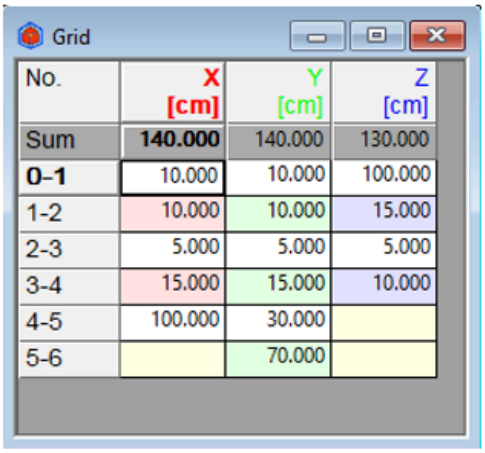
In the **Notes dialog box** (via *Edit → Notes...*) some comments can be written for the current document, e.g. project description, operator, date. These notes are saved in the .vtr file.



B.5 Grid window

Menu command: *Edit → Grid*

The **Grid window** is the input window for grid mesh sizes of a rectangular 3D grid.





The grid

The three orthogonal directions of the grid correspond to the axes of the coordinate system.

This is either XYZ (a right-handed Cartesian coordinate system) or RCL (consisting of rows, columns and layers, as in a 3D matrix). Use the commands *Settings → Show XYZ* and *Settings → Show RCL* to switch between both systems (see A.9). Always choose the Z axis (or the layer direction) along the physical vertical direction (because otherwise the function *Image → Rotate* works unnaturally).

The positions of the object blocks (cf. section B.8) are expressed in integer numbers referring to the sequence numbers of the grid planes in the three main directions.

- In XYZ the grid plane positions are numbered 0,1,2,...
- In RCL the grid plane positions are numbered 1,3,5,... (the even numbers refer to space intervals between two consecutive grid planes).

The intersection points of three orthogonal grid planes inside or at the surface of material blocks are nodes of the system with unknown temperature.

Grid meshes

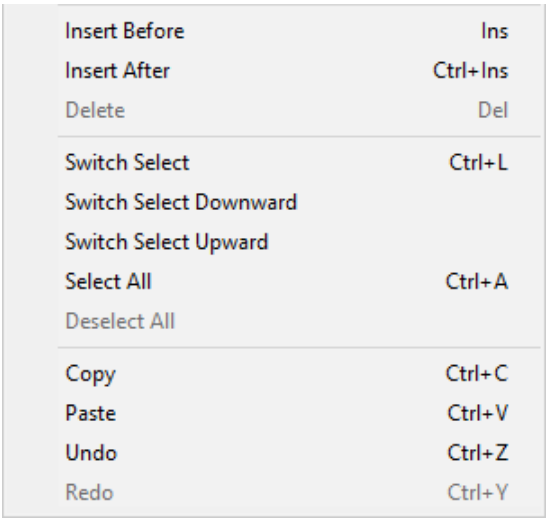
The meshes, or distances between successive grid planes in any of the three axis directions, are editable in the **Grid window**.

The background colours of the grid meshes editable in the **Grid window** alternate, indicating the underlying minimum grid (cf. section B.7).

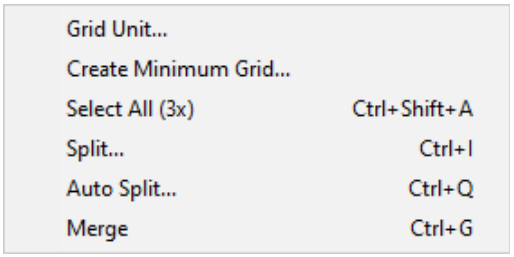
The grid unit is set in the dialog box opened with the command *Grid → Grid Unit...* (see B.6.3).

B.6 Grid functions

Lower part of Edit submenu



Grid submenu



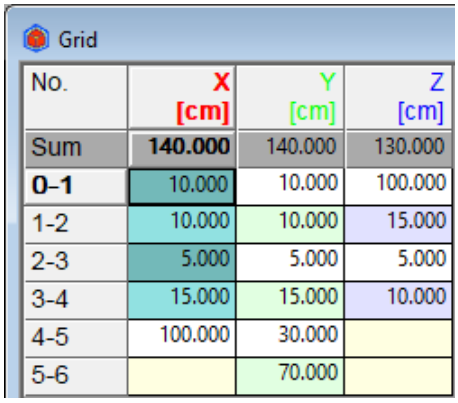
B.6.1 Insert mesh

- *Edit → Insert Before*
  Insert a new mesh (with size = 1 grid unit) before the active mesh.
- *Edit → Insert After*
  Insert a new mesh (with size = 1 grid unit) after the active mesh.

B.6.2 Grid mesh selections

Individual grid meshes can be selected to perform global actions (split, merge).

The selected meshes are marked in turquoise (light or dark dependent on the underlying minimum grid).


Grid

No.	X [cm]	Y [cm]	Z [cm]
Sum	140.000	140.000	130.000
0-1	10.000	10.000	100.000
1-2	10.000	10.000	15.000
2-3	5.000	5.000	5.000
3-4	15.000	15.000	10.000
4-5	100.000	30.000	
5-6		70.000	

Meshes can be selected by clicking (left mouse button) the cell while holding the <Ctrl> key. A selection range can be defined by holding the <Shift> key. All meshes within the same column ΔX or ΔY can by selected by clicking (left mouse button) the column header.

Alternatively, menu functions can be used to select meshes.

From the Edit submenu:

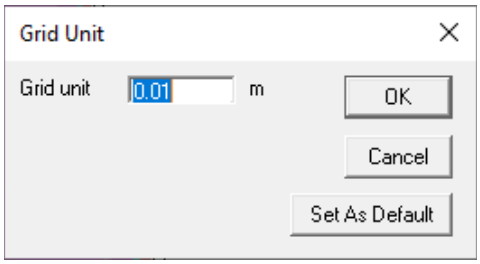
- *Edit → Switch Select*
  Switch on/off the selection for the active mesh.
- *Edit → Switch Select Downward*
  Switch on/off the selection of the active mesh and all next meshes (on the same axis) with equal selection mode.
- *Edit → Switch Select Upward*
  Switch on/off the selection of the active mesh and all previous meshes (on the same axis) with equal selection mode.
- *Edit → Select All*
  Select all meshes on the same axis as the active mesh.
- *Edit → Deselect All*
  Deselect all meshes (on all three axes).

From the Grid submenu:

- *Grid → Select All (3x)*
  Select all meshes (on all three axes).

B.6.3 Grid unit

Menu command: *Grid → Grid Unit...*

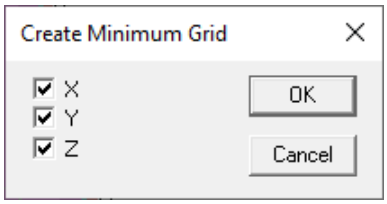




The grid unit can be set in the dialog box. When the unit is altered, the user has the option to transform the grid widths, i.e. to keep the same physical dimensions of the model, or to keep the grid widths and thus scale the model according to the change in grid unit.

B.6.4 Create minimum grid

Menu command: *Grid → Create Minimum Grid...*



Creates a minimum grid in the indicated directions.

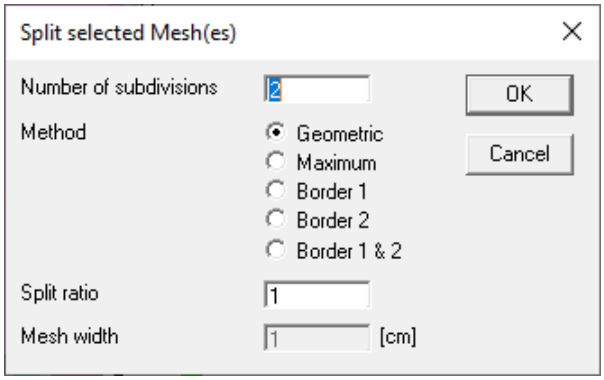
All grid coordinates (and grid meshes) are rounded off to three decimal places.

The meshes of the minimum grid only pass through borders of blocks (as defined in the **Blocks window**) and points in which node boundary conditions are defined (in the **Node BCs window**).

The functions *Blocks → Transform Coordinates...* and *Blocks → Insert File...* (see B.8) automatically create a minimum grid in all three axis directions.

B.6.5 Split (selected) mesh(es)

Menu command: *Grid → Split...*



Split either the selected meshes or the active mesh (when no selection is defined) into smaller parts according to the given method.

Method: Geometric

Each mesh is split into the specified number of subdivisions using a geometric series (i.e. a series with a constant ratio between successive terms) with specified split ratio.

- When the split ratio = 1, then all created submeshes are equal.
- When the split ratio < 1, then the submeshes have decreasing mesh widths.
- When the split ratio > 1, then the submeshes have increasing mesh widths.

  A negative or zero split ratio is not allowed.

Method: Maximum

All submeshes have a constant mesh width (as defined by the parameter Mesh width in the dialog box), except for the last submesh, for which the width equals the remainder after division. This last submesh can however never be smaller than 0.1 % of the maximum mesh width.

Method: Border 1

Each mesh is split into two parts, when the original mesh size is greater than the given mesh width (parameter Mesh width in dialog box). The first submesh (border 1) gets the size of the given mesh width, the second submesh gets the size of the remainder.

Method: Border 2

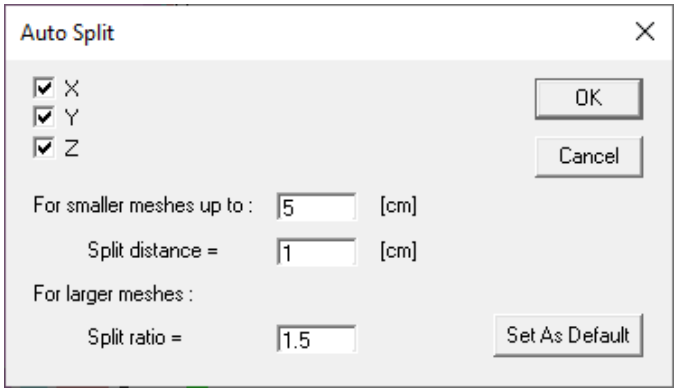
Each mesh is split into two parts, when the original mesh size is greater than the given mesh width (parameter Mesh width in dialog box). The second submesh (border 2) gets the size of the given mesh width, the first submesh gets the size of the remainder.

Method: Border 1 & 2

Calls successively: split using border 2, then split using border 1.
So each mesh is split into three parts (when the original mesh size is large enough): both outer submeshes get the size of the given mesh width, the middle submesh gets the size of the remainder.

B.6.6 Auto split

Menu command: *Grid → Auto Split...*



Split all meshes in the specified directions (X, Y, Z) into smaller parts.

For originally smaller meshes (up to a given mesh size), the mesh is split into submeshes having a given split distance, and possibly a last submesh with width equal to the remainder after division (cf. Method: Maximum in section B.6.5).

The other (larger) meshes are split using a geometric subdivision using a given split ratio (cf. Method: Geometric in section B.6.5) starting with the split distance used for smaller meshes at both ends of the original mesh.

The function "Auto Split" is intended to achieve a reliable grid refinement suitable for calculation using only one function call (with well-chosen parameters in the dialog box) starting from the minimum grid.

B.6.7 Merge selected meshes

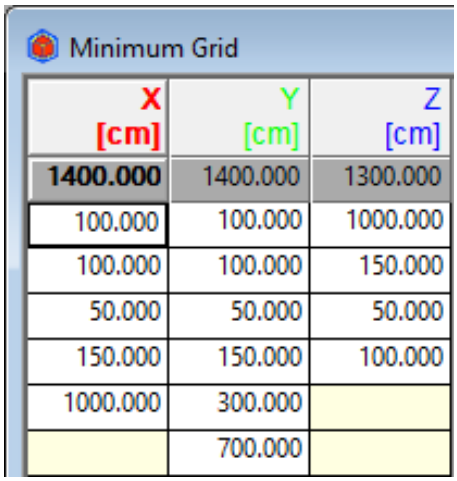
Menu command: *Grid → Merge*

Merge each group of consecutive selected meshes into larger meshes (which width equals the sum of the widths of the original meshes). Meshes at both sides of a block border (or a node BC) cannot be merged. The grid planes of the minimum grid are always preserved.

B.7 Minimum Grid window

Menu command: *Edit → Minimum Grid*

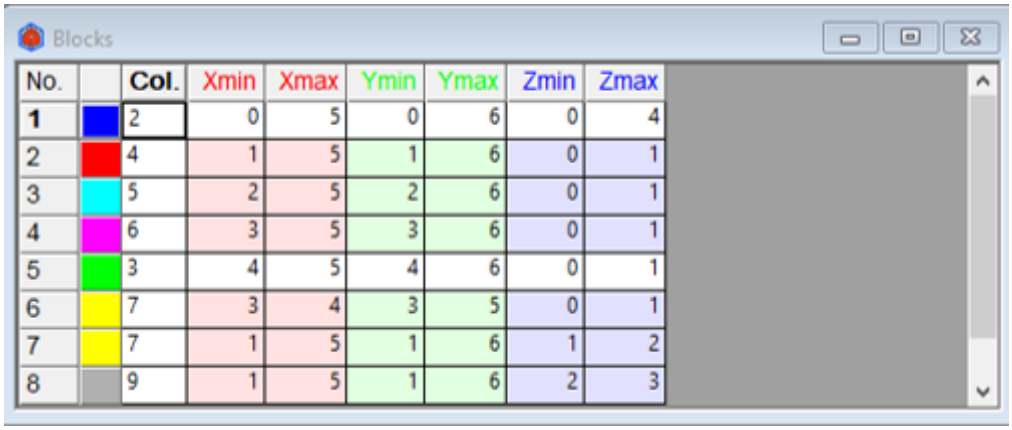
The **Minimum Grid window** shows the meshes of the minimum grid (through block borders and node BCs) without touching the actual grid (defined in the **Grid window**). This is thus not to be confused with creating a minimum grid (see B.6.4).

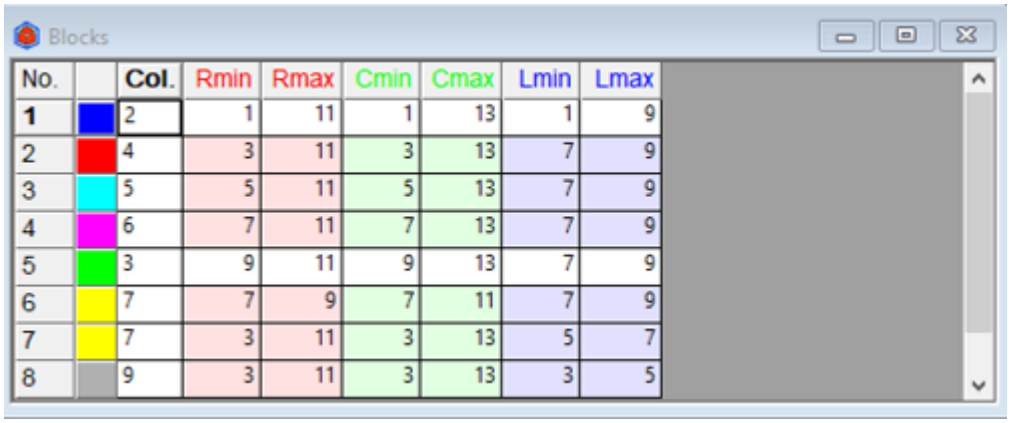


When a mesh size in the **Minimum Grid window** is edited, all underlying meshes of the **Grid window** are resized proportionally.

B.8 Blocks window

Menu command: *Edit → Blocks*





The object geometry is defined by blocks in the **Blocks window**.

Each block has the following parameters:

Colour number (between 0 and 255) referring to the **Colours window**.

The parameter "type" in the **Colours window** determines whether the block is massive or hollow.

- A *material* block (colour type MATERIAL, EQUIMAT or TRANSMAT) is massive and is added to the object geometry.
- A *boundary condition* block (colour type BC_SIMPL, BC_FRE_S, BC_FREE, BC_SKY or BC_NOSKY) is hollow and is subtracted from the object geometry.

6 coordinates of block borders (i.e. minimum and maximum coordinate per axis).

The coordinates are grid plane numbers, which positions are defined in the **Grid window**.

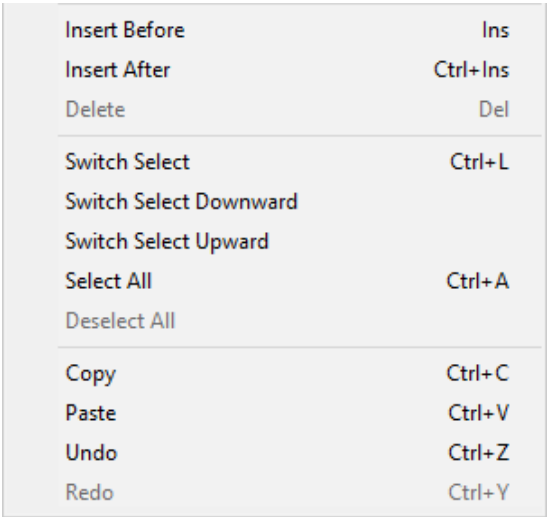
In the coordinate system XYZ the grid plane numbers are integers starting from 0.
In the coordinate system RCL the grid plane numbers are odd numbers starting from 1.
The background colour is light red, light green or light blue for a material block, and white for a boundary condition block.

The size of the active block (width x depth x height) is visible in the status bar.

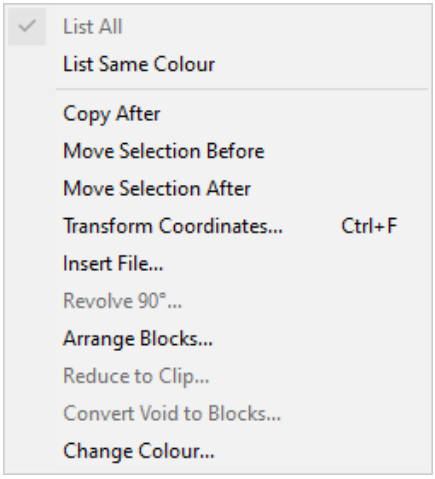
Blocks with higher sequence number have priority over (i.e. overwrite the common parts of) blocks with lower sequence number. This is called the superposition rule.

B.9 Block functions

Lower part of Edit submenu



Blocks submenu



B.9.1 Insert block

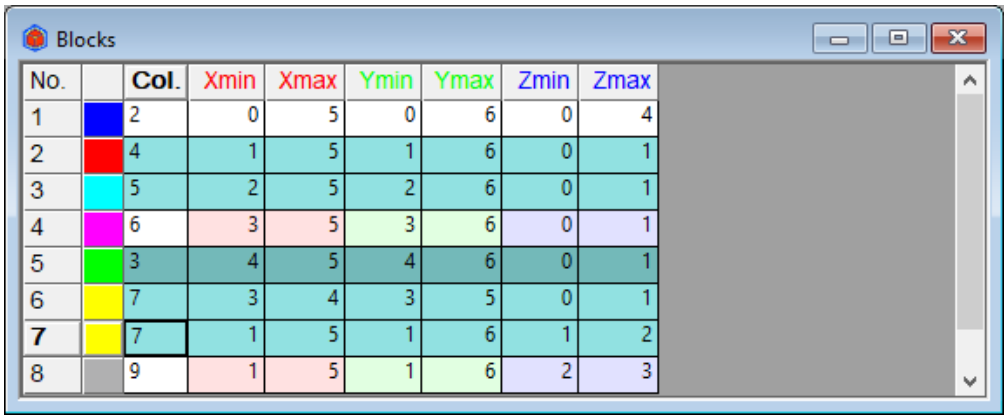
- *Edit → Insert Before*
  Insert a new block before the active block.
- *Edit → Insert After*
  Insert a new block after the active block.

The new block gets the following initial values:

- The colour of the last edited block colour.
- Block coordinates:
  - In XYZ: Xmin = 0, Xmax = 1, Ymin = 0, Ymax = 1, Zmin = 0, Zmax = 1.

- In RCL: Rmin = 1, Rmax = 3, Cmin = 1, Cmax = 3, Lmin = 1, Lmax = 3.

B.9.2 Block selections



No.		Col.	Xmin	Xmax	Ymin	Ymax	Zmin	Zmax
1		2	0	5	0	6	0	4
2		4	1	5	1	6	0	1
3		5	2	5	2	6	0	1
4		6	3	5	3	6	0	1
5		3	4	5	4	6	0	1
6		7	3	4	3	5	0	1
7		7	1	5	1	6	1	2
8		9	1	5	1	6	2	3

Individual blocks can be selected to perform global actions (delete, copy, move, transform).

The selected blocks are marked in (light or dark) turquoise.

Blocks can be selected by clicking (left mouse button) the row number. Holding the <Shift> and <Ctrl> keys, a selection range can be defined.

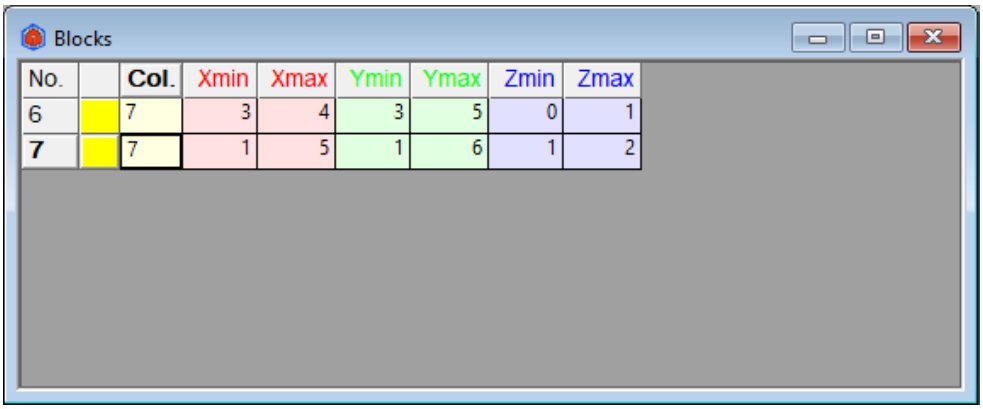
Alternatively, menu functions can be used to select blocks.

Block selection functions:

- *Edit → Switch Select*
  Switch on/off the selection for the active block.
- *Edit → Switch Select Downward*
  Switch on/off the selection of the active block and all next blocks with equal selection mode.
- *Edit → Switch Select Upward*
  Switch on/off the selection of the active block and all previous blocks with equal selection mode.
- *Edit → Select All*
  Select all blocks.
- *Edit → Deselect All*
  Deselect all blocks.

A useful function to make block selections is the view option:

- *Blocks → List Same Colour*
  This function only shows in the **Blocks window** all blocks having the same colour as the active block.



  - The block sequence numbers are the original sequence numbers (as in the normal view option *Blocks → List All*).
  - The colour number cannot be edited.
  - The block coordinates can be edited.
  - Block selection is possible (but restricted to the displayed blocks, e.g. for *Edit → Select All*).
  - All other block functions are disabled.
    To enable these functions again, return to *Blocks → List All*. (All active block selections remain preserved.)

- *Blocks → List All*
  List all blocks again, to terminate the view option *Blocks → List Same Colour*.

B.9.3 Delete, copy, move

The following functions have no effect on the active grid (as defined in the **Grid window**).

- *Edit → Delete*
  Delete selected block(s), otherwise (in absence of a block selection) delete the active block.

- *Blocks → Copy After*
  Copy selected block(s) to a new position in the blocks list.
  If there is no block selection, the active block is copied after itself.
  If there is a block selection, the selected blocks are copied immediately after the active block.
  After the copy operation the copied blocks become the new block selection.

- *Blocks → Move Selection Before*
  Move selected block(s) to a new position in the blocks list.
  If there is no block selection, this function is disabled
  If there is a block selection, the selected blocks are moved immediately before the active block.
  After the move operation the selected blocks remain selected.

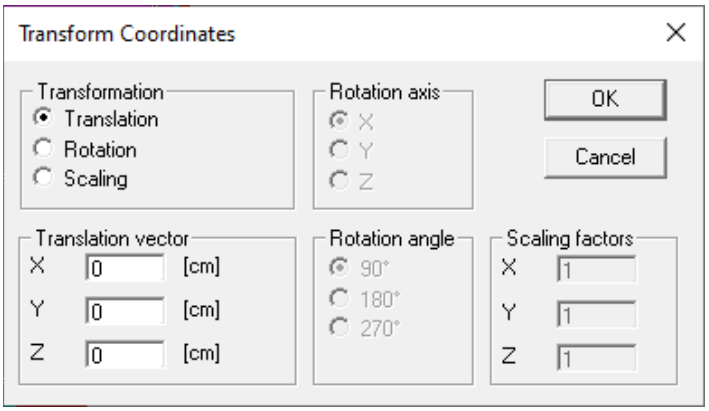
- *Blocks → Move Selection After*
  Move selected block(s) to a new position in the blocks list.
  If there is no block selection, this function is disabled
  If there is a block selection, the selected blocks are moved immediately after the active

block.
After the move operation the selected blocks remain selected.

B.9.4 Transform; insert file

The following functions create new coordinates, and so have an effect on the current grid. After the block operation, a minimum grid is automatically created (with grid planes through the block borders).

- *Blocks → Transform Coordinates...*
  Transform selected block(s), otherwise (in absence of a block selection) transform the active block.



There are three types of transformations:

- Translation: over a given translation vector.
- Rotation: over a right angle (90°, 180° or 270°) around a rotation axis parallel to X, Y or Z through the current origin (*Coord → Set Origin*, see C.12).
- Scaling: using given scaling factors.
  The origin (*Coord → Set Origin*, see C.12) is the centre of the scaling.
  A mirror transformation is possible using a negative scaling factor.
  E.g. mirror X corresponds to the scaling factors –1, 1, 1.

- *Blocks → Insert File...*
  The blocks of a VOLTRA or TRISCO data file (extensions .vtr or .trc) are inserted after the active block.
  The new blocks are positioned from the current origin (*Coord → Set Origin*, see C.12).
  Colour definitions for not yet present colours (in the **Colours window**) are also read from the imported data file.
  After the operation the inserted blocks are selected.

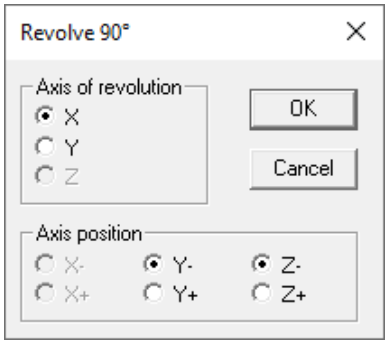
B.9.5 Revolve

Menu command: *Blocks → Revolve 90°...*

Only enabled for a 2D geometry (i.e. with only 1 mesh in X, Y or Z).

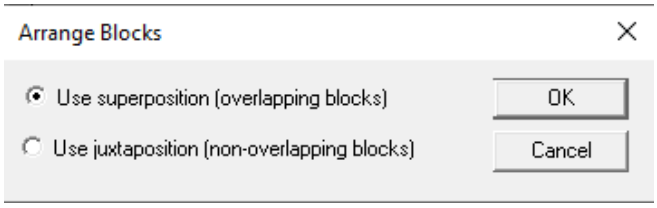
Turns the 2D geometry around an axis parallel to X, Y or Z over 90° to form a 3D corner joint with the 2D geometry as profile section. This corresponds to a rectangular revolution around the axis. The axis lies at the negative or positive end of the bounding box of the 2D geometry (specified as axis position X- or X+, Y- or Y+, Z- or Z+ in the dialog box).

The grid is extended in the third dimension (with originally 1 mesh) as a copy of the grid meshes in the direction perpendicular to the axis of revolution.



B.9.6 Arrange blocks

Menu command: *Blocks → Arrange Blocks...*



This function recalculates the list of blocks in the **Blocks window** based on the final object geometry and boundary conditions geometry in the **Image window** (viewed with *Image → Show Materials* and *Image → Show BCs*). The grid data in the **Grid window** remain unchanged.

The blocks are generated using an algorithm that tries to minimise the number of blocks.

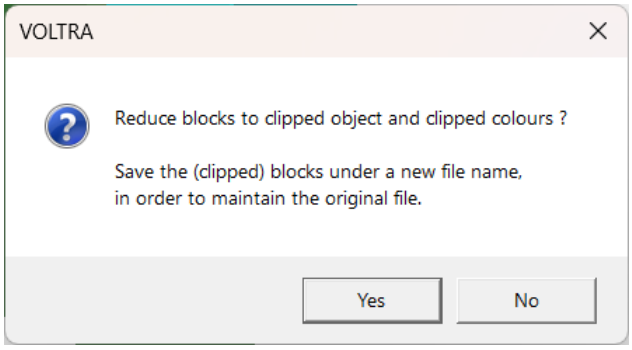
The new list of blocks may be longer than the original list of blocks.

There are two options:

- Superposition of blocks: with possible overlaps of successive blocks in order to minimise the number of new blocks.
- Juxtapostion of blocks: all new blocks are positioned adjacent to one another, without any overlaps of blocks.

B.9.7 Reduce blocks to clipped object and colours

Menu command: *Blocks → Reduce to Clip...*



The answer “No” cancels the operation. “Yes” confirms the operation.

The blocks are recalculated (using block arrangement with superposition; cf. section B.9.6) for the object and boundary condition geometry (as visible in the **Image window**) restricted to the active object clipping (cf. section C.10) and colours clipping (cf. section C.11).

Afterwards clip object and clip colours are reset.

A minimum grid is automatically created (with grid planes through the new block borders).

The node boundary conditions, borderface boundary conditions, material boundary conditions, areas in **Areas window**, and U values in **U Values window** are cleared.

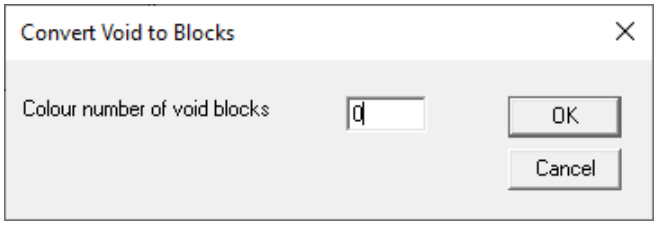
B.9.8 Detect void (adiabatic) spaces within object

Menu command: *Blocks → Convert Void to Blocks...*

This function is only enabled if there are void spaces (not covered by object or boundary condition blocks) present in the 3D grid space. Void spaces are not directly visible in the **Image window**. Void spaces should normally be avoided in the grid space, because they imply implicitly adiabatic boundary conditions for all adjacent object surfaces.

This function allows to make them visible in the **Image window** and convert the voids to blocks with a certain colour number.

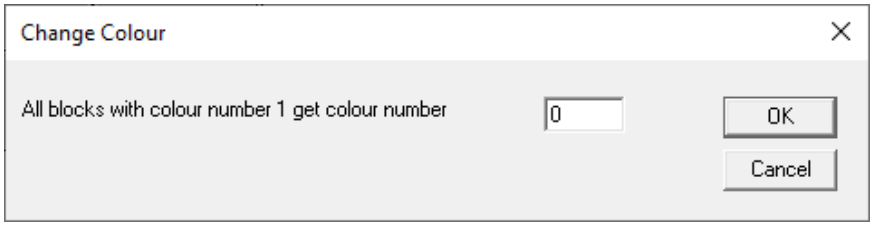
It is advisable to choose a colour number not yet present in the model (B.11.1).



All void spaces in the 3D grid space are converted to blocks having the chosen (new) colour number. All blocks are then recalculated using block arrangement with superposition (cf. section B.9.6). If the active cell is moved to the new colour in the **Colours window** (use type = MATERIAL), the outline of the void spaces becomes visible in the **Image window**.

B.9.9 Redefine colour globally

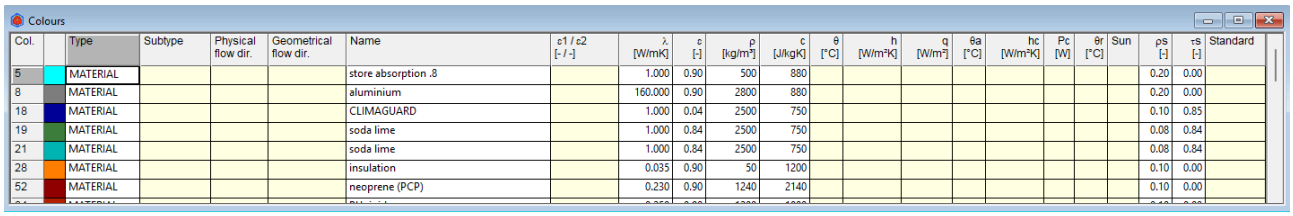
Menu command: *Blocks → Change Colour...*



All blocks having the same colour number as the active block (with the active cell in the **Blocks window**) get a new colour number (as defined in the dialog box).

B.10 Colours window

Menu command: *Edit → Colours*



The **Colours window** defines of the colours associated to the blocks in the **Blocks window**.

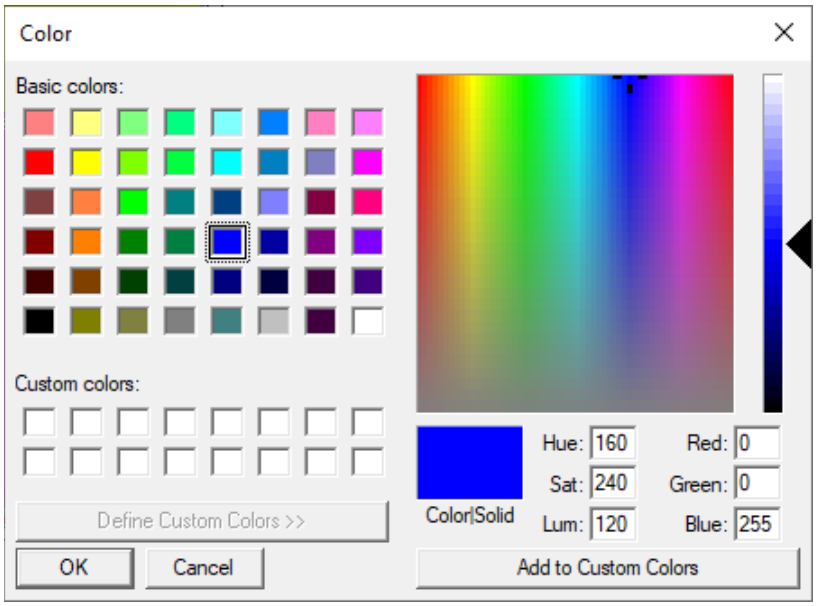
Via the colours the thermal properties of the blocks are set.

B.10.1 Colour number

Each colour is identified by a colour number or colour index: an integer between 0 and 255.

Thus a maximum of 256 colours can occur in a VOLTRA data file.

The fill colour belonging to each colour number can be changed. Position the active cell (surrounded by thick black lines) onto the second column (showing the actual colour), and double click (with the left mouse button) or press Enter. Choose a new colour in the standard Colour dialog box.



When VOLTRA is started up, the default colour values are loaded from the file MatColor.rgb.

This file contains the red, green and blue components (as numbers between 0 and 255) for all 256 colour numbers.

The (edited) colours for a specific VOLTRA data file are stored in the file data (as RGB values), and are automatically loaded when the file is read.

B.10.2 Colour type

Each colour stands for a material or a surface boundary condition dependent on the colour type.

The following types are always available (with or without the RADCON module):

- **MATERIAL**: material defined by thermal conductivity.

- **EQUIMAT**: equivalent material defined by a single equivalent thermal conductivity (e.g. air cavity). The equivalent conductivity is typically calculated from standard rules (D.1.1).
- **BC_SIMPL**: simplified surface boundary condition, defined by a global surface heat transfer coefficient (for combined convection and radiation), environmental temperature and possible additional surface heat flux.
- **BC_FRE_S**: enclosure with unknown (free-floating) internal temperature, possible additional internal heat power, simplified surface heat transfer defined by a global surface heat transfer coefficient and possible additional surface heat flux.

With the RADCON module (which supposes separate convection and radiation based on view factors, see Chapter I) also:

- **BC_FREE**: surface boundary condition modelled as enclosure with unknown (free-floating) air temperature, radiation between the surfaces (using the infrared emissivities of the adjacent materials) and convective heat transfer coefficient.
- **BC_SKY**: surface boundary condition modelled as an environment with defined air and radiation temperature (the sky) and convective heat transfer coefficient.
- **BC_NOSKY**: surface boundary condition modelled as an environment with defined air temperature, no radiation to the environment (no sky) and convective heat transfer coefficient.
- **TRANSMAT**: transparent material defined by thermal conductivity, and radiation between the surfaces (using the infrared emissivities of the adjacent materials).

B.10.3 Standard (last column)

Standard used for the automatic calculation of λ_{eq} or h_c (cf. section D.1).

- NIHIL: no automatic calculation of thermal properties.
- EN10077: standard EN ISO 10077-2.
- EN6946: standard EN ISO 6946.

Within each standard different subtypes can be chosen.

B.10.4 Colour subtype

Subtype toggle value to define automatic calculation of equivalent thermal conductivity (type EQUIMAT and TRANSMAT) or convective heat transfer coefficient (BC_FREE, BC_SKY and BC_NOSKY). The accessible subtypes are dependent on the actual type (3rd column) and standard (last column).

For type EQUIMAT:

- NIHIL: user defined thermal conductivity (no automatic calculation).
- CAVITY: calculated equivalent thermal conductivity for unventilated air cavity (EN10077 or EN6946).
- CAVITY_E: calculated equivalent thermal conductivity for slightly ventilated air cavity (EN10077).
- LAYER: calculated equivalent thermal conductivity for unventilated air layer (EN6946).

For type BC_SIMPL:

- NIHIL: user defined global heat surface coefficient.
- HI_NORML: normal internal heat transfer (EN10077).
- HI_REDUC: reduced internal heat transfer (EN10077).
- HI: internal heat transfer (EN6946).
- HE: external heat transfer (EN10077 or EN6946).
- HI_CAV_E: slightly ventilated air cavity at the internal surface (EN10077 – radiosity method).
- HE_CAV_E: slightly ventilated air cavity at the external surface (EN10077 – radiosity method).

For type BC_FREE:

- NIHIL: user defined convective heat transfer coefficient (no automatic calculation).
- CAVITY: calculated convective heat transfer coefficient for unventilated air cavity.
- CONVEC: calculated convective heat transfer coefficient for large air space.

For type BC_SKY or BC_NOSKY:

- NIHIL: user defined convective heat transfer coefficient (no automatic calculation).
- CONVEC: calculated convective heat transfer coefficient for natural convection.

For type TRANSMAT:

- NIHIL: user defined thermal conductivity (no automatic calculation).
- CAVITY: calculated equivalent thermal conductivity for unventilated air cavity.

For more details on the corresponding calculation procedures see D.1.

B.10.5 Colour physical heat flow direction

Heat flow direction with reference to the physical reality:

- HOR: horizontal heat flow.
- UP: upward heat flow.
- DOWN: downward heat flow.
- ANY: user-defined heat flow direction (by defining the inclination angle of the construction element and assuming a heat flow perpendicular to the construction element)

B.10.6 Colour geometric heat flow direction

Heat flow direction with reference to the grid XYZ coordinate system (see B.5):

- X: heat flow parallel to the X axis.
- Y: heat flow parallel to the Y axis.
- Z: heat flow parallel to the Z axis.
- Xy: heat flow parallel to the X axis and cavity width parallel to the Y axis.
- Xz: heat flow parallel to the X axis and cavity width parallel to the Z axis.
- Yx: heat flow parallel to the Y axis and cavity width parallel to the X axis.

- Yz: heat flow parallel to the Y axis and cavity width parallel to the Z axis.
- Zx: heat flow parallel to the Z axis and cavity width parallel to the X axis.
- Zy: heat flow parallel to the Z axis and cavity width parallel to the Y axis.

B.10.7 Colour name

Enter a name for the material or boundary condition. The name should not exceed 25 characters.

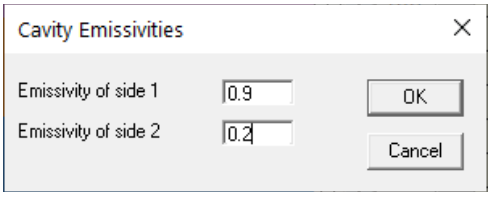
The status bar shows the volume of all blocks having this colour.

B.10.8 Physical properties

The relevant thermal properties for each colour type (that are used in a calculation) are shown in the **Colours window**. Values on a white background can be edited. Values on a yellow background are automatically calculated. Empty cells refer to irrelevant properties for the active colour type.

Thermal properties:

- ε1 / ε2 [- / -]: infrared emissivities (values between 0.001 and 1) at warm and cold sides of cavity (type EQUIMAT).
  Editing opens a dialog box for definition of the values.



  If any emissivity differs from 0.9 (the default value), the corresponding grid cell in the **Colours window** is highlighted in blue.
- λ [W/(m.K)]: thermal conductivity of material (MATERIAL), equivalent material (EQUIMAT) or transparent material (TRANSMAT).
- ε [-]: infrared emissivity (value between 0 and 1) of material surfaces (MATERIAL, EQUIMAT) adjacent to boundary condition or cavity with radiative heat transfer.
- ρ [kg/m^3]: density of material (MATERIAL), equivalent material (EQUIMAT), transparent material (TRANSMAT) or enclosure (BC_FREE and BC_FRE_S).
- c [J/(kg.K)]: specific heat of material (MATERIAL), equivalent material (EQUIMAT), transparent material (TRANSMAT) or enclosure (BC_FREE and BC_FRE_S).
- θ [°C]: temperature of simplified boundary condition (BC_SIMPL).
- h [W/(m^2.K)]: global surface heat transfer coefficient (BC_SIMPL, BC_FRE_S).
- q [W/m^2]: heat flux dissipated onto adjacent material surfaces (BC_SIMPL, BC_FRE_S, BC_FREE, BC_SKY, BC_NOSKY).
- θa [°C]: air temperature (BC_SKY, BC_NOSKY).
- hc [W/(m^2.K)]: convective heat transfer coefficient (BC_FREE, BC_SKY, BC_NOSKY).
- Pc [W]: convective power dissipated in the environment (BC_FRE_S, BC_FREE).

- θr [°C]: radiation temperature (BC_SKY).

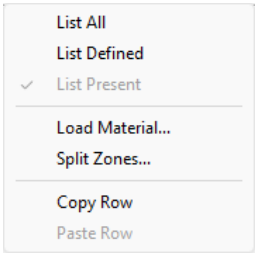
Solar data (four columns with header Sun, ρs, Specular and τs):

- Sun: sun active status (YES or NO) to define a solar zone, for a colour of type BC_SIMPL or BC_SKY.
- ρs [-]: solar reflection factor (for a colour of type MATERIAL), a constant value (between 0 and 1), or a function reference (R##) when the solar reflection factor is a function of the angle of incidence (see also section B.12.2). See D.9.1 for a detailed explanation of the definition of ρs.
- τs [-]: solar transmission factor (for a colour of type MATERIAL), a number between 0 and 1 representing the resulting transmission factor – after reflection - of the MATERIAL layer and all transparent material layers (MATERIAL or EQUIMAT) behind it until another zone or an opaque material (τs equals 0) is reached. See D.9.1 for a detailed explanation of the definition of τs.

The properties λ, c, θ, h, q, θa, hc, Pc, θr and ρs can each either have a constant value (as in TRISCO) or refer to a function (defined in the **Functions window**) via the function reference name: L## (for λ), C## (for c), T## (for θ, θa, θr), H## (for h, hc), I## (for q), P## (for Pc), R## (for ρs).

B.11 Colour functions

The colour functions are available through the **Colours submenu**.



B.11.1 List colours

Colours → List All

List all available 256 colours.

The edit fields for colour numbers not referenced from the **Blocks window** are disabled (grey text).
The list is useful in choosing new colour numbers for blocks in the **Blocks window**.

Colours → List Defined

List all defined colours.
This option displays besides all present colours (i.e. referenced from the **Blocks window**), the colour definitions from the colour database and from files previously opened in the current VOLTRA session, which are still present in memory.
The edit fields for colour numbers not referenced from the **Blocks window** are disabled (grey text).

Colours → List Present

List all colours referenced from the **Blocks window**.

This is the default setting when a new project file is opened.

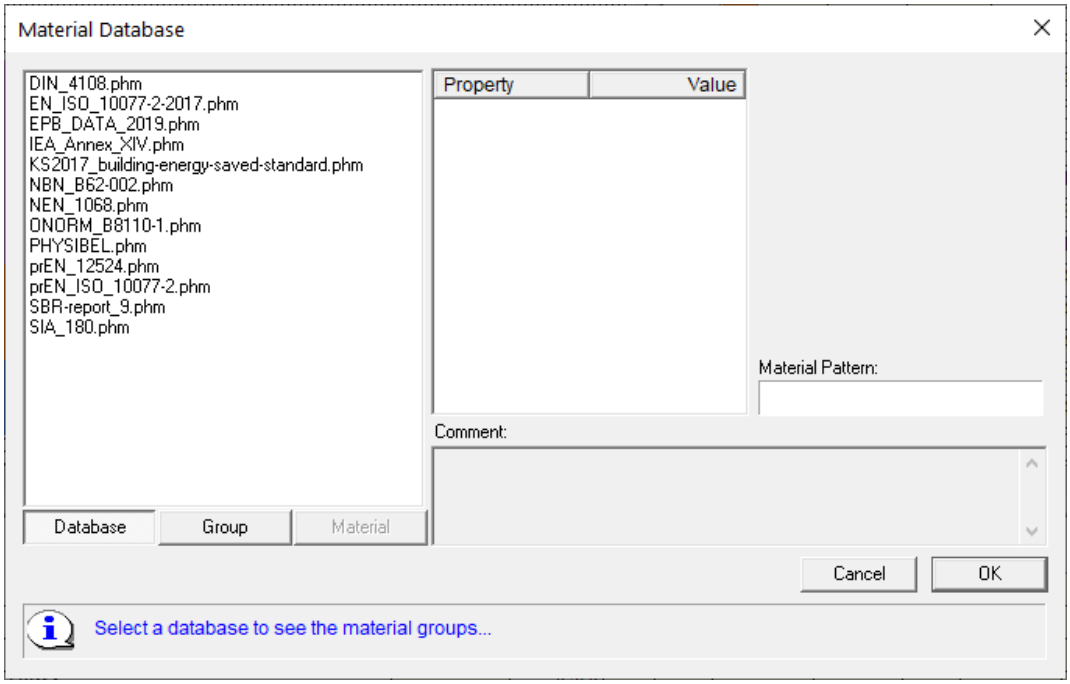
B.11.2 Load material properties

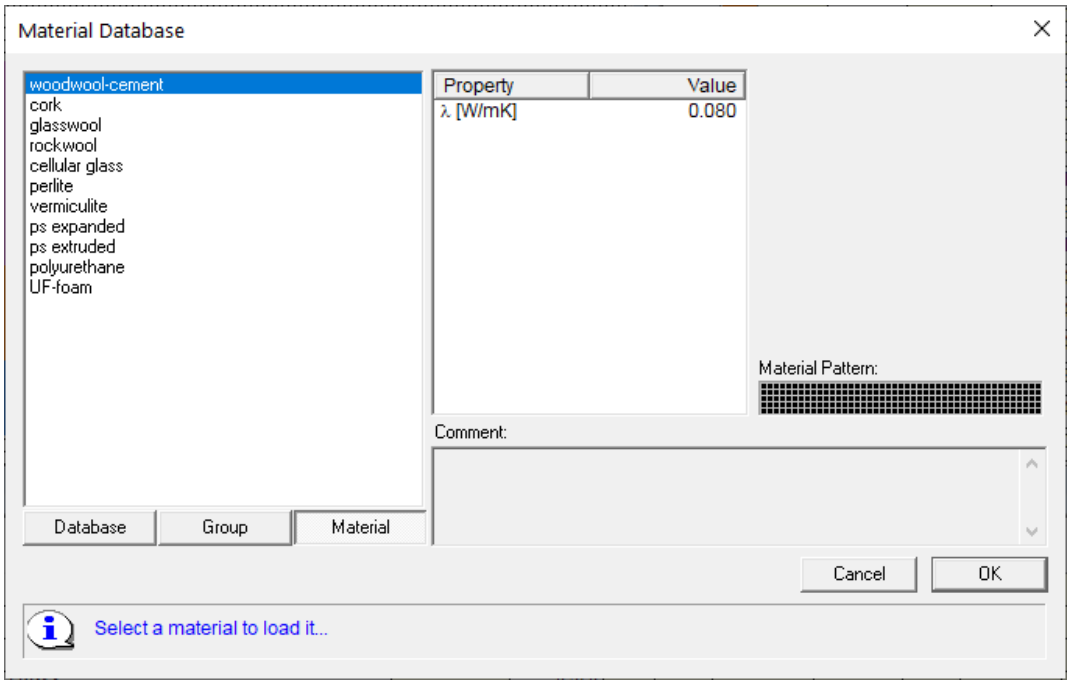
Menu command: *Colours → Load Material...*

The Material Database dialog box allows to select a material from a group of materials in a selected database.

The available databases contain the thermal conductivities of different materials according to several European standards, and are stored in the files with extension .phm in the data folder ...\Physibel\Materials (see A.6). The files can be edited using the program MatEdit.exe in the same folder.

After selecting a material and pressing the OK button, the corresponding properties are loaded for the active colour number.





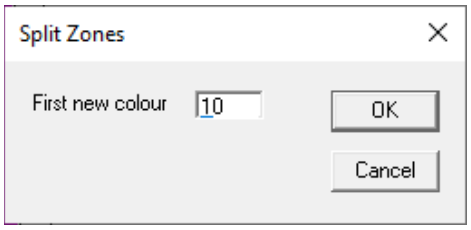
B.11.3 Split zones

Menu command: *Colours → Split Zones...*

(Only enabled from the active **Colours window**)

Split all blocks of the active colour (i.e. where the cursor is located in the **Colours window**) into blocks of different new colours, so that each new colour represents a single zone (i.e. united and not separated by any other colour).

The first new colour must be entered in the dialog box.



Only colours not yet present in the **Colours window** are used.

The new colours get the thermal properties from the original active colour.

The original active colour is not reused as new colour.

A warning appears when there are not enough free colours for all zones.

The function is useful for colours (e.g. of type BC_FRE_S, BC_FREE, TRANSMAT) which require single-zone colours to be valid in the calculation model.

B.11.4 Copy - paste

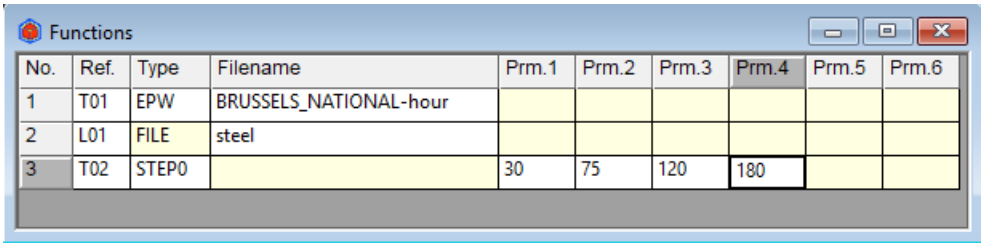
The functions *Colours → Copy Row* and *Colours → Paste Row* allow to copy all properties from one colour number to another colour number.

B.12 Functions window

For the variable boundary conditions and material properties VOLTRA refers to functions: either an internal function (parameters defined in the **Functions window**), or an external function (data read from a file).

All functions that are used in a VOLTRA data file, are defined in the **Functions window**, opened with the command *Edit → Functions*.

The picture below shows an example of **Functions window**.



Each function is uniquely defined by a function reference name:

Reference	Description	Units	Date file formats
T01 up to T99	Temperature function	[°C]	*.fte, *.epw, *.csv
H01 up to H99	Surface heat transfer coefficient function	[W/(m^2.K)]	*.fht
P01 up to P99	Power function	[W]	*.ffh
I01 up to I99	Imposed flux function (I01 is used in the Solar Data dialog box)	[W/m^2]	*.fir
G01 up to G99	Horizontal global solar radiation function (only G01 is used in the Solar Data dialog box)	[W/m^2]	*.fsg, *.epw, *.csv
D01 up to D99	Horizontal diffuse solar radiation function (only D01 is used in the Solar Data dialog box)	[W/m^2]	*.fsd, *.epw, *.csv
V01 up to V99	ventilation function	[m^3/s]	*.fvr
E01 up to E99	Emissivity function	[-]	*.fem
M01 up to M99	thermal conductivity function (time-dependent)	[W/(m.K)]	*.fco
L01 up to L99	thermal conductivity function (temperature dependent)	[W/(m.K)]	*.fla
C01 up to C99	specific heat function (temperature dependent)	[J/(kg.K)]	*.fce
R01 up to R99	solar reflection factor function (angular dependent)	[-]	*.frf
S01 up to S99	Solar transmission factor function	[-]	*.ftr

Function reference names can be used in:
the **Colours window** (columns with header λ, ε, c, θ, h, q, θa, hc, Pc, θr, ρs, τs),
the **Node BCs window** (columns with header θ and P),
the **Borderface BCs window** (columns with header θ and q),

the **Material BCs window** (columns with header θ and p)
the **Ventilation Flows window** (column with header "Flow"),
The **Controls window** (columns with header "θ target", "Function from", "Function to").

Temperature functions (L## and C##) and solar reflection factor functions (R##) can only have type FILE (see B.12.2).

All function values are used in VOLTRA as average values over the simulation time step, where the simulation time is understood as the middle of the simulation time step (see D.9.2).

A new function can be inserted via *Edit → Insert Row*.

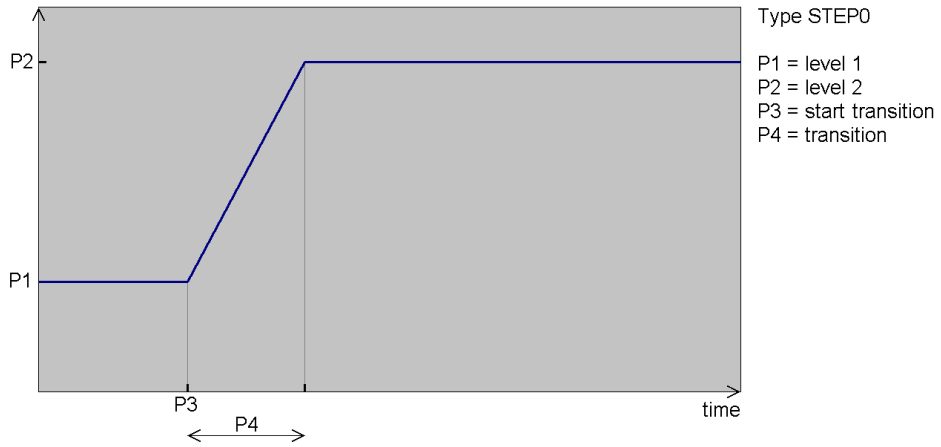
The current function can be deleted via *Edit → Delete Row*.

B.12.1 Internal time functions (types STEP0, STEP1, STEP2, SINE, and SINPOS)

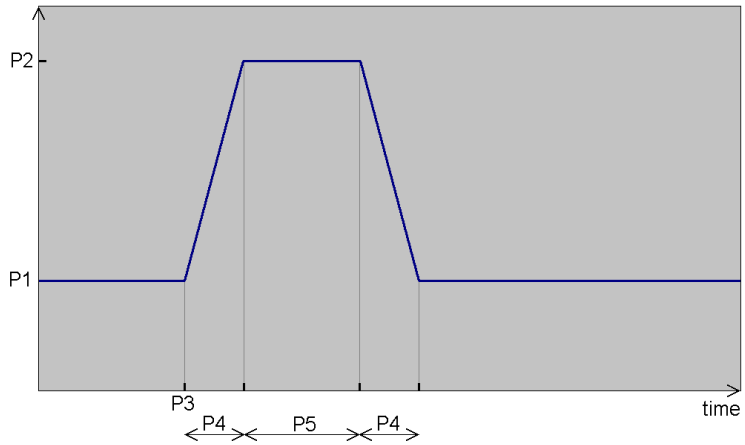
An internal time function is a built-in program function, defined by one or more parameters. VOLTRA contains 5 built-in function templates.

The meaning of each parameter per function type is explained in the status bar (cf. figures below).
All time parameters must be entered in minutes (e.g. 1 day = 24 h x 60 min/h = 1440 min).

- STEP0 for a step function defined by 4 parameters (P1, P2, P3, P4)



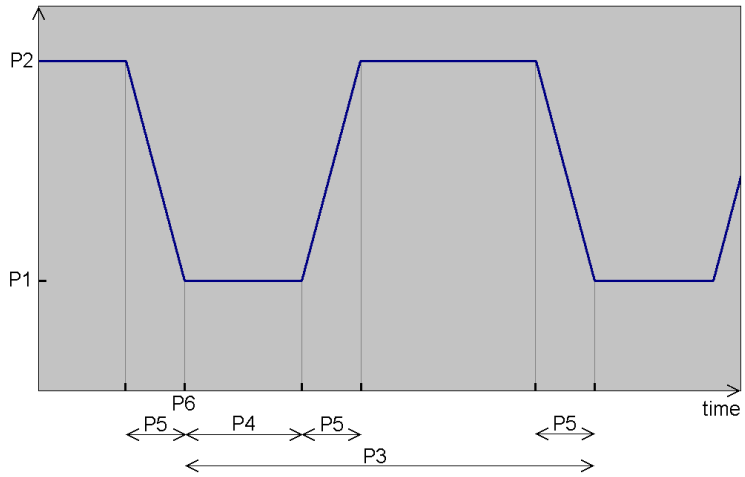
- STEP1 for a step function defined by 5 parameters (P1, P2, P3, P4, P5)



Type STEP1

P1 = level 1
P2 = level 2
P3 = start transition
P4 = transition
P5 = duration level 2

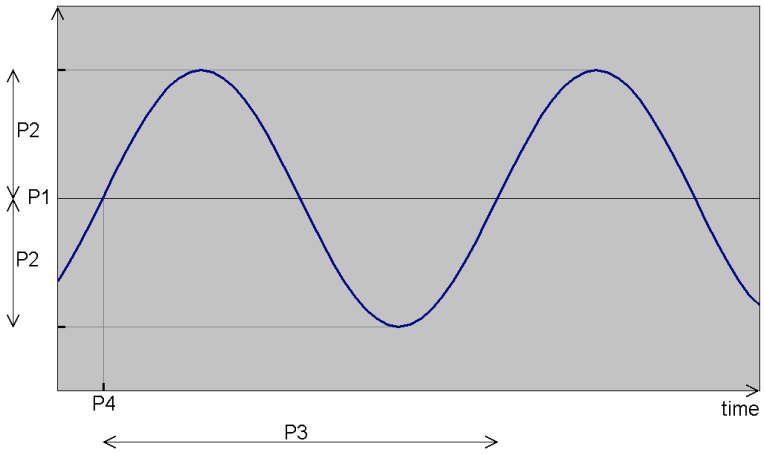
- STEP2 for a step function defined by 6 parameters (P1, P2, P3, P4, P5, P6)



Type STEP2

P1 = level 1
P2 = level 2
P3 = period
P4 = duration level 1
P5 = transition
P6 = start level 1

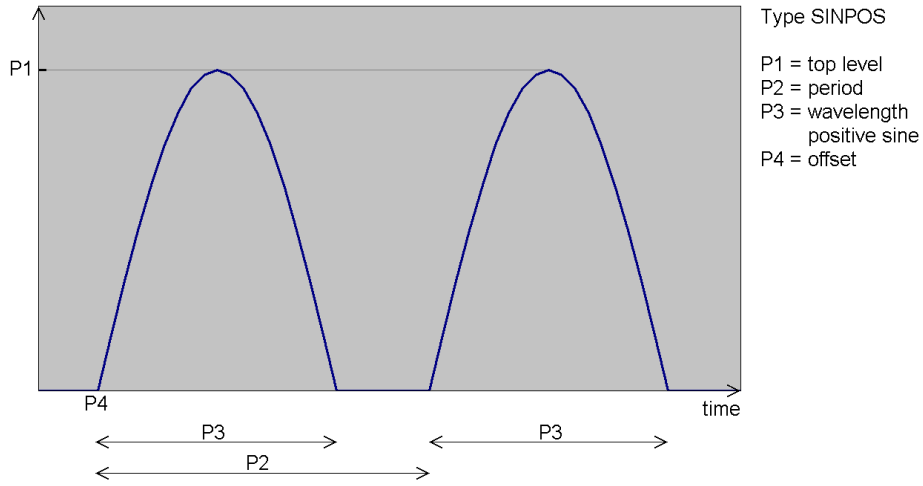
- SINE for a sine function, defined by 4 parameters (P1, P2, P3, P4)



Type SINE

P1 = mean value
P2 = amplitude
P3 = period
P4 = offset

- SINPOS for a positive sine function, defined by 4 parameters (P1, P2, P3, P4)



B.12.2 External functions (type FILE)

An external time function is defined in a separate data file, with extension:

- .fte (temperature function)
- .fht (surface heat transfer coefficient function)
- .ffh (power function)
- .fir (imposed flux function)
- .fsg (horizontal global solar radiation function)
- .fsd (horizontal diffuse solar radiation function)
- .fvr (ventilation function)
- .fco (time-dependent thermal conductivity function)
- .fem (time-dependent emissivity)
- .ftr (time-dependent solar transmission factor).

The program FUNCEDIT (cf. Chapter H) allows to edit these files.

External data as a source for an external time function can either be formatted as a linear function, or as a step function (see H.3).

In case of a *linear function format*, function values at simulation times in between external data timestamps are obtained by simple linear interpolation between the external data timestamps.

In case of a *step function format*, the values in the external data file are interpreted as the average value during the data interval preceding the external data timestamp and are thus assumed constant and equal to the value at the data interval's end timestamp for any simulation times in between external data timestamps[1]. An exception is made for solar

[1]Note that in older versions of VOLTRA (v8.1 and before), step function formatted external data were interpreted differently, namely as the average value during the data interval *following* the external data timestamp and thus assumed constant and equal to the value at the data interval's *begin* timestamp for any simulation times in between external data timestamps.

radiation functions (G## and D##), in which case external data formatted as a step function are converted to instantaneous values at simulation times using a mathematical algorithm (see D.9.2)[2]. For this algorithm to work, the simulation start time and time step have to be defined in such a way that the simulation hits the hours precisely (see D.2.1).

Some material properties (thermal conductivity, specific heat, solar reflection factor) may be dependent on another property (temperature, angle of incidence).

These temperature functions (thermal conductivity, specific heat) or angular functions (solar reflection factor) are defined in separate data files, with extension:

- .fla (temperature dependent thermal conductivity function)
- .fce (temperature dependent specific heat function)
- .frf (angular dependent solar reflection factor function).

These function data files are text files consisting of two or more text lines:

x0 y0
x1 y1
x2 y2
...
xn yn

with $x0 \leq x1 \leq x2 \leq ... \leq xn$.

Each text line consists of two float numbers, the abscissa (i.e. a temperature value or an angle of incidence) and the ordinate (i.e. the corresponding thermal conductivity, specific heat value or solar reflection factor) of a characteristic point of the function. The function is linearly interpolated between the characteristic points. If $x < x0$, then $y = y0$. If $x > xn$, then $y = yn$.

The user can define these functions with a text editor (e.g. Windows Notepad).

When the function filename field is double clicked (or when Enter is pressed), a file path can be chosen in the Open dialog box. If the external files are stored in the same directory (or a subdirectory of this directory) as the VOLTRA data file, the file paths are relative (so that copying of the .vtr file together with the function files to another directory has the desired result that the new .vtr file refers to the function files in the same new data directory). If the external files are not stored in the same directory as the VOLTRA data file, the absolute file paths are saved in the .vtr file.

B.12.3 Weather data files (types EPW and TMY3)

A weather data file is a data file containing observations of weather variables such as temperature, relative humidity, solar radiation, wind speed, etc. for a specific location. Two different data file formats can be imported into VOLTRA: EPW or *EnergyPlus Weather* (extension .epw), and TMY3 or *Typical Meteorological Year* (extension .csv).

The weather data files can only be used as functions in VOLTRA if they (i) contain data for a complete year, (ii) are defined in local time (not solar time), (iii) do not contain lines with

[2] This algorithm was not included in older versions VOLTRA (v8.1 and before).

more than 512 characters, and (iv) start at hour 1 after midnight, January 1st.[3]

Weather data files can be used as source for the functions:

- Temperature (T##): the dry-bulb temperature of the weather file is read and interpreted as instantaneous values occurring at the timestamps denoted in the weather data file.
- Global horizontal radiation (G##): the total horizontal radiation is read and interpreted as the total amount of radiation (Wh/m^2) received in the interval preceding each timestamp.
- Horizontal diffuse radiation (D##): the diffuse horizontal radiation is read and interpreted as the total amount of diffuse radiation (Wh/m^2) received in the interval preceding each timestamp.

The temperature values from the weather data file are linearly interpolated between the instantaneous values occurring at the data file timestamps[4]. The radiation values from the weather data file are converted to instantaneous values at simulation times using a mathematical algorithm (see D.9.2). For this algorithm to work, the simulation start time and time step have to be defined in such a way that the simulation hits the hours precisely (see D.2.1).

If function G01 for global horizontal radiation is a weather data file, the geographical location is automatically read and used in the solar processor (see B.18.1).

When the function filename field is double clicked (or when Enter is pressed), a file path can be chosen in the Open dialog box.

If the weather data files are stored in the same directory (or a subdirectory of this directory) as the VOLTRA data file, the file paths are relative (so that copying of the .vtr file together with the weather data files to another directory has the desired result that the new .vtr file refers to the weather data files in the same new data directory). If the weather data files are not stored in the same directory as the VOLTRA data file, the absolute file paths are saved in the .vtr file.

B.13 Function Graph window

The **Function Graph window** shows a graph of the current function in the **Functions window**.

The **Function Graph window** is automatically opened when the **Functions window** is activated, and automatically closed when another window is activated (e.g. **Measures window** or **Colours window**).

Zoom functions (via the Zoom submenu) allow to view function details.

The quantities defined by the X axis (time, temperature, angle of incidence) and Y axis (temperature, surface heat transfer coefficient, flux, power, lambda, emissivity, specific heat, solar transmission factor, solar reflection factor) are determined by the current function in the **Functions window**.

[3] In case the simulation start time is hour 0 (midnight), January 1st, the values from the weather data file for hour 1 are copied for hour 0.

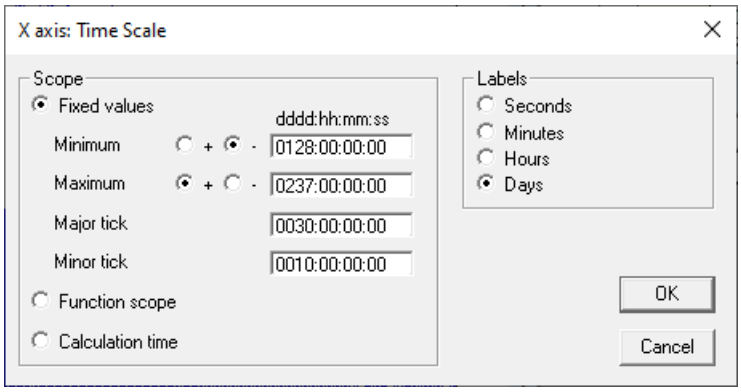
[4] Note that in the .epw file format valid values for the 'hour' field run from 1 to 24 (hour 1 is 00:01 to 01:00 in local time) and values for the 'minute' field run from 1 to 60. This means that e.g. the combination of hour 2 and minute 60 indicates a time stamp of 02:00 in local time. If the 'minute' field is undefined in the .epw file (typically with value 0), it is interpreted as having value 60.

The origin of the X axis of a time function (i.e. time = 0) corresponds to the calculation start time, as defined in the **Calculation Parameters dialog box**.

In case of solar radiation functions (G## and D##) using an external data file with data formatted as a step function (B.12.2) or a weather file (B.12.3) the source data are visualised in the **Function Graph window**, not the converted function values used in the simulation (see D.9.2).

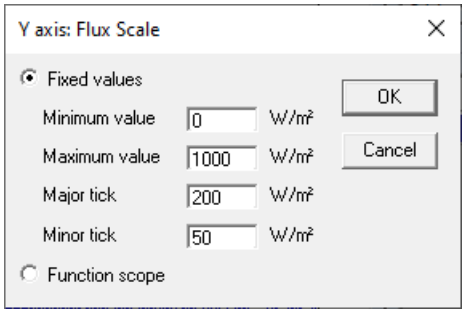
B.14 Function commands

The X and Y axis tick marks and labels from the graph shown in the **Function Graph window** can be redefined via the commands *Functions → X Scale...* and *Functions → Y Scale...*.



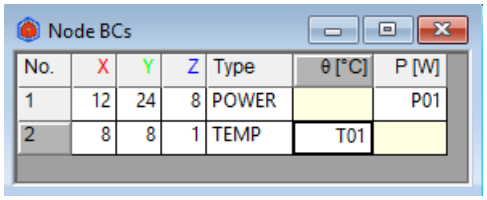
The scope of the time scale on the X axis can be chosen between fixed values, function scope (corresponding to the definition of the function) and calculation time (corresponding to the calculation duration as defined in the **Calculation Parameters dialog box**).

When fixed values are chosen, the minimum and maximum value (with plus or minus sign), and the major and minor tick distances can be set. Also the label unit can be chosen (seconds, minutes, hours or days).



B.15 Node BCs window

Menu command: *Edit → Node BCs*



A node boundary condition ("node BC") is a grid node inside the object (defined by three grid number coordinates) with either a temperature value (type TEMP) or a heat power

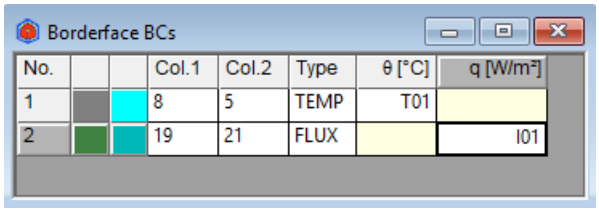
(type POWER). Both can be a numeric value (for a constant property) or can refer to a function (defined in the **Functions window**), T## or P##.

Add a new node BC using *Edit → Insert Before* or *Edit → Insert After*.

Delete a node BC using *Edit → Delete*.

B.16 Borderface BCs window

Menu command: *Edit → Borderface BCs*



A border face boundary condition ("borderface BC") imposes either a temperature (type TEMP) or a heat flux (type FLUX) in all interfaces between object blocks (in the final object composition as shown in the **Image window**) having different fixed colours.

The position of the interfaces for the active borderface BC is highlighted in the **Image window** (if the option *Image → Hide Outline* is not active).

Edit fields

- Col.1: colour number (between 0 and 255) of object blocks at one side of the interface.
- Col.2: colour number (between 0 and 255) of object blocks at the other side of the interface.
- Type: TEMP or FLUX.
- θ: temperature (in °C), numeric value (for a constant property) or can refer to a function T## (defined in the **Functions window**).
- q: heat flux (in W/m^2), numeric value (for a constant property) or can refer to a function I## (defined in the **Functions window**).

Remarks

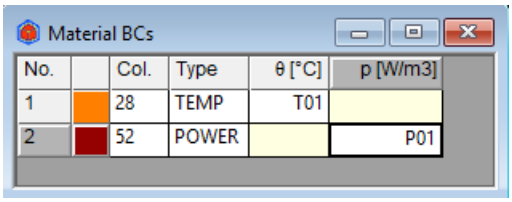
- When a colour number is entered that does not occur in the **Colours window**, then this input is not accepted (the input field is cleared).
- To be valid at least one of both colour numbers must reference a material (type MATERIAL, EQUIMAT or TRANSMAT).

Edit commands

- *Edit → Insert After*: add a new borderface BC.
- *Edit → Delete*: delete a borderface BC.

B.17 Material BCs window

Menu command: *Edit → Material BCs*



A material boundary condition ("material BC") imposes either a temperature (type TEMP) or a heat power density (type POWER) in all object blocks (in the final object composition as shown in the **Image window**) having a given colour number, either as constant values or as functions of time (T## or P##).

The outline of the object blocks related to the active material BC are highlighted in the **Image window** (if the option *Image → Hide Outline* is not active).

Edit commands

- *Edit > Insert After*: add a new material BC.
- *Edit > Delete*: delete a material BC.

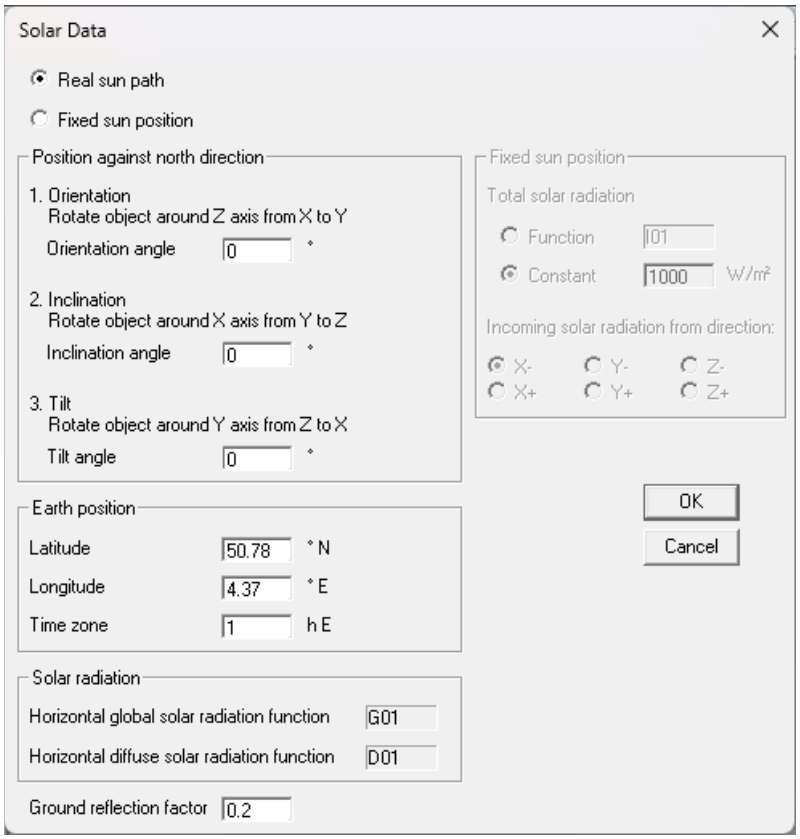
Remark

A drawback of this kind of boundary condition is that the total heat flow into all object blocks of the given colour number cannot be output (in the **Text window**).

B.18 Solar data

The solar processor requires the definition of parameters in the Solar Data dialog box (*Edit → Solar Data...*).

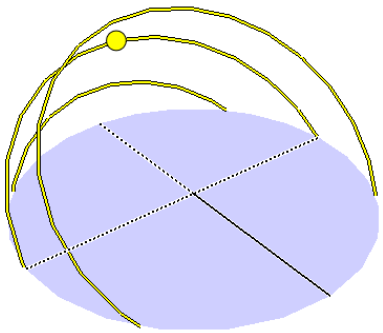
First, a choice can be made between a Real sun path or a simplified calculation with a Fixed sun position.



B.18.1 Real sun path

North orientation[5]: The modelled object must be positioned into the horizontal coordinate system of the physical reality (see C.8) by defining three rotation angles.

If all three angles are set to 0°, the object's XYZ coordinate system (as shown in the bottom left corner of the **Image window**) coincides exactly with the horizontal coordinate system, i.e. the X axis points to the north, the Y axis points to the west and the Z axis points to the zenith.



The three angles to be defined by the user position the object by rotating the object around it's own coordinate axes. As a visual aid, the object's position is updated in the **Image window** (as with Show North activated, see C.8).

- The first rotation is around the Z-axis.
  This rotation happens within the object's XY plane (i.e. a rotation around the object's Z axis).

[5]Note that the definition of these angles has changed since version 9.0.

The object is rotated counterclockwise. A rotation angle of 90°, for example, points the object's X-axis to the west orientation.

- The second rotation is around its X-axis to define the object's inclination.
- The third rotation is around its Y-axis to define the object's tilt.

Geographical location ("earth position"): latitude (°N), longitude (°E) and time zone (h E). The input fields are disabled when these parameters are present in the weather data file containing the global solar radiation data, as defined in the **Functions window** (G01). The coordinates are then loaded from the solar radiation weather data file.

Solar radiation functions: the function reference names of the horizontal global and diffuse solar radiation functions are always G01 and D01. These solar radiation functions must be defined in the **Functions window**.

Ground reflection factor: reflection factor of (horizontal) ground surface around the object model. The reflected solar radiation is projected on the external object surfaces using an isotropic diffuse model.
The input value must be between 0 and 1. A typical value is 0.2. For very reflective grounds a higher value can be used (e.g. snow: 0.8).

B.18.2 Fixed sun position

An alternative to a Real sun path is to impose a fixed sun position for a simplified calculation. In this case, the user can choose between a function or a constant value for the total solar radiation. The total solar radiation is treated as direct radiation in the calculation (see D.9.2).

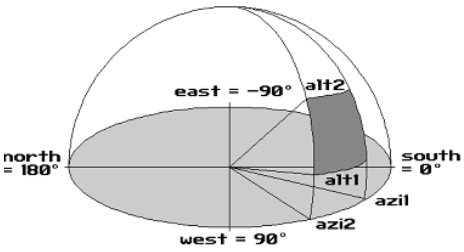
In case a function is selected, the function reference name I01 is always used and should be defined in the **Functions window**. This function I01 contains time-dependent data for the total solar radiation.

The user can select one of six directions from which the radiation originates, namely along the main coordinate axes.

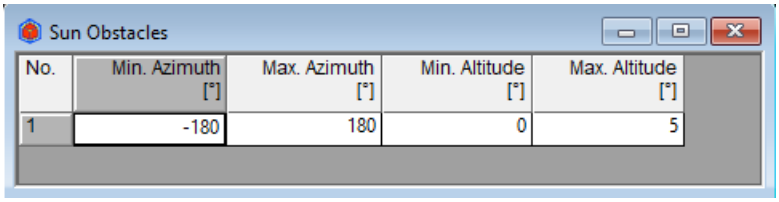
B.19 Sun obstacles window

Sun obstacles around the 3D object are defined in the **Sun Obstacles window**, opened with *Edit → Sun Obstacles*. Each sun obstacle is a spherical rectangle defined by azimuth and altitude coordinates on the sky hemisphere within which the sun is invisible.

The azimuth and altitude coordinates are defined in the horizontal coordinate system (see B.18.1 and B.15). Each sun obstacle is defined by four parameters: minimum and maximum azimuth (clockwise angle between –180° and 180°, 0° = south, 90° = west), and minimum and maximum altitude (between 0° at the horizon and 90° at the zenith).



By default, one sun obstacle (defined by a zone up to 5° high above the horizon in all directions around) is present for each new VOLTRA file (cf. figure below).



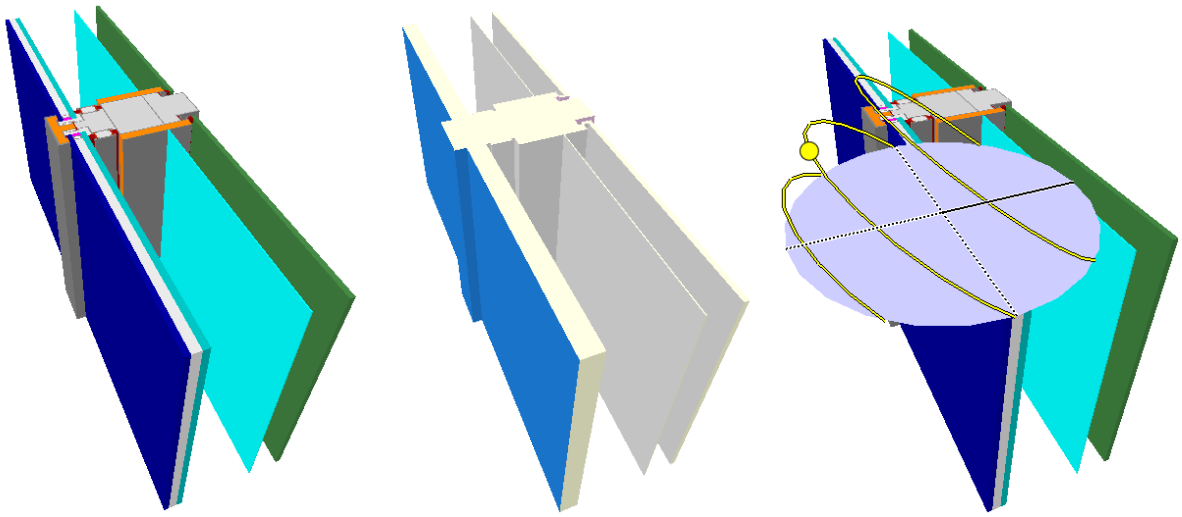
A new sun obstacle is added using *Edit → Insert Row*.
A sun obstacle is deleted using *Edit → Delete Row*.

Note that the **Sun Obstacles window** is disabled in case a fixed sun position is defined in the Solar Data dialog box (see B.18), as sun obstacles are not taken into account in that case.

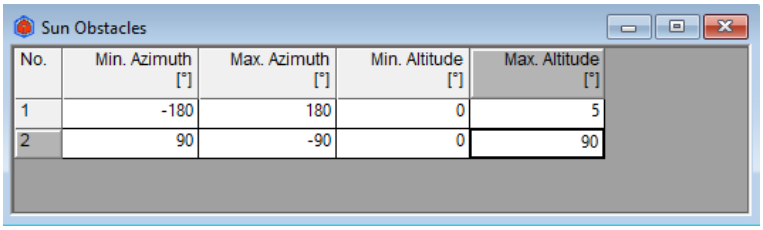
Remark

If the solar zone does not surround the complete object, the explicit definition of sun obstacles is necessary to prevent that the sun hits the object from unwanted angles.

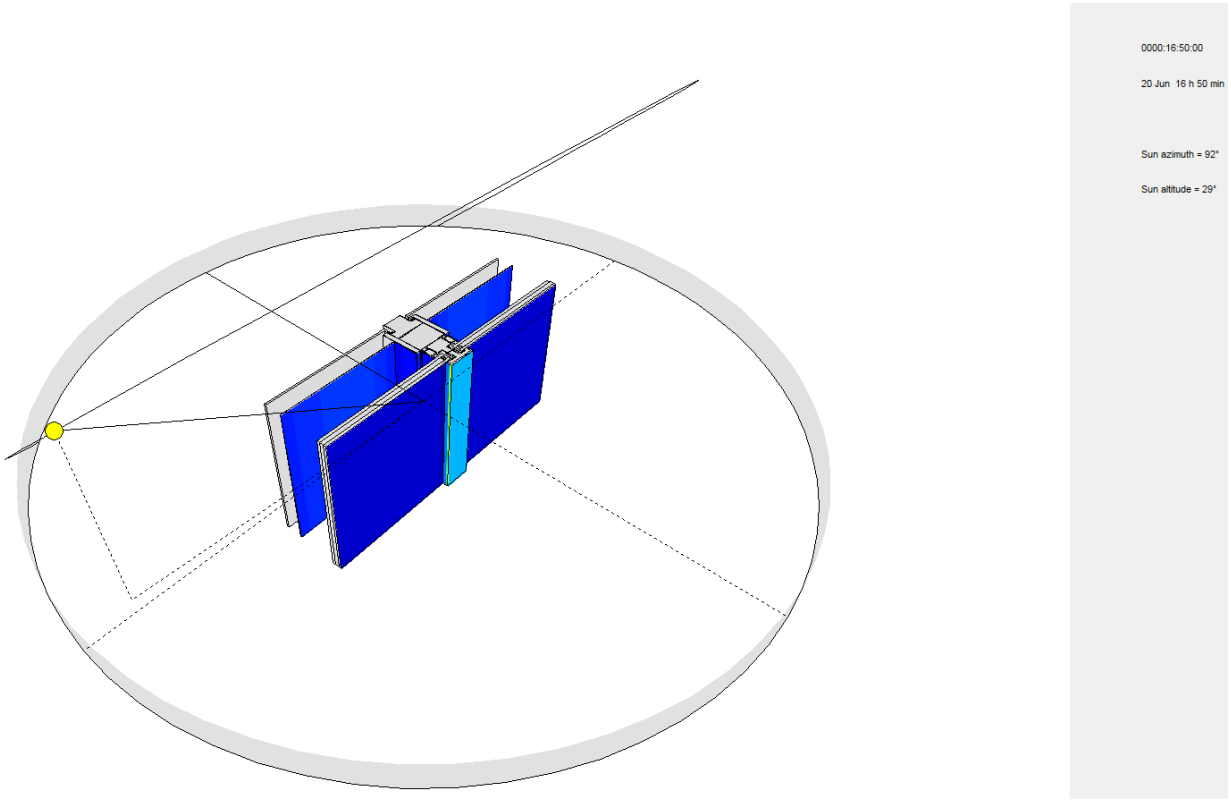
An example is shown below. The object represents part of a double façade, facing south.

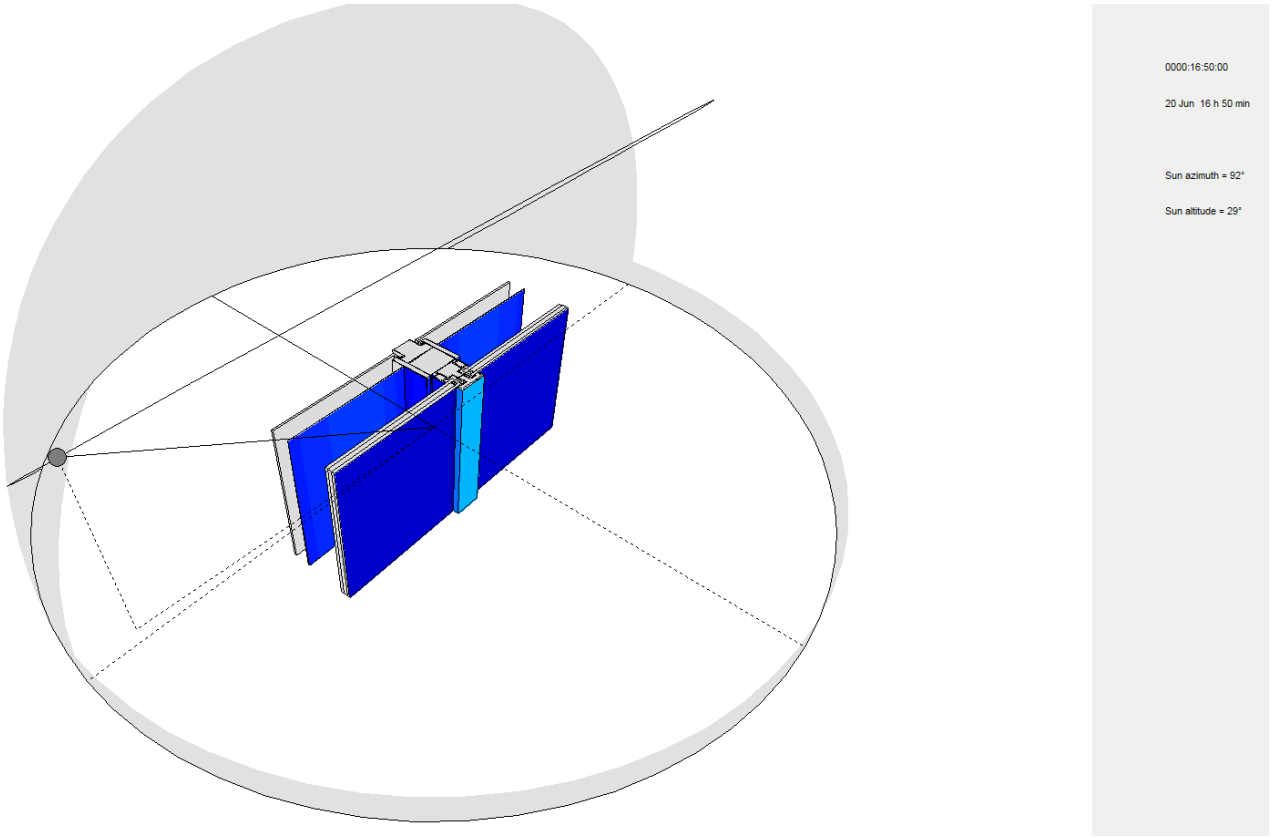


In order to avoid unwanted solar radiation on the mullion when the sun would actually be behind the building on which this façade is installed, solar obstacles have to be defined.



The first figure below shows unwanted solar radiation due to the lack of solar obstacles. The second figure shows the desired effect.





B.20 Controls window

VOLTRA includes temperature controls mimicking the working of a thermostat, defined in the **Controls window**. A control decides between the application of two functions based on a sensor temperature, target temperature and purpose.

Two functions, used as boundary condition or material property function, need to be defined in the **Functions window** and referred to in the **Colours window**, **Node BCs window**, **Borderface BCs window**, **Material BCs window** or **Ventilation Flows window**.

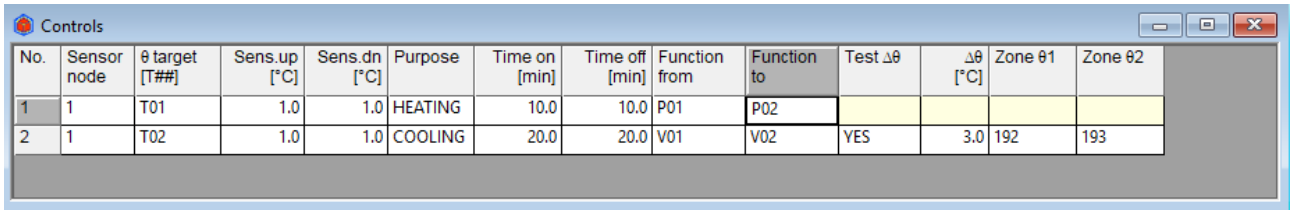
The sensor temperature refers to the temperature at the output node specified in the **Output Nodes window**. At each time step, this value is compared to the target temperature to determine whether the control should be activated or deactivated.

A time dependent target temperature for heating or cooling is imposed. These controls thus allow e.g. applying a higher power or ventilation flow if heating or cooling is required.

To avoid rapid fluctuations of this on/off system, a hysteresis control can be defined using the upward and downward thermostat sensitivities.

Moreover, a minimum operational duration of control action after activation or deactivation can be defined via time on (minimum time (in minutes) after control action is activated) or time off (minimum time (in minutes) after control action is deactivated) parameters.

A new control can be added using *Edit → Insert Before* or *Edit → Insert After*, and deleted with *Edit → Delete*.



No.	Sensor node	θ target [T##]	Sens.up [°C]	Sens.dn [°C]	Purpose	Time on [min]	Time off [min]	Function from	Function to	Test Δθ	Δθ [°C]	Zone θ1	Zone θ2
1	1	T01	1.0	1.0	HEATING	10.0	10.0	P01	P02				
2	1	T02	1.0	1.0	COOLING	20.0	20.0	V01	V02	YES	3.0	192	193

A control is defined by the following elements (corresponding to the different columns):

- Sensor node, where at each calculation time step the temperature is compared to the desired target temperature. The sensor node is defined by the reference number of an output node in the **Output Nodes window** (cf. section B.21).
- Target temperature: function reference of a temperature function (T01 up to T99) defined in the **Functions window** (cf. section B.12).
- Thermostat sensitivity upward [°C]:
  When a heating control is active, a necessary condition to switch off the control is that the sensor temperature is greater than the target temperature plus the upward sensitivity.
  When a cooling control is not active, a necessary condition to switch on the control is that the sensor temperature is greater than the target temperature plus the upward sensitivity.
- Thermostat sensitivity downward [°C]:
  When a heating control is not active, a necessary condition to switch on the control is that the sensor temperature is less than the target temperature minus the downward sensitivity.
  When a cooling control is active, a necessary condition to switch off the control is that the sensor temperature is less than the target temperature minus the downward sensitivity.
- Purpose: a control can be intended for either heating or cooling.
- Minimum time on: minimum duration that the control must continuously remain switched on. A control will be checked and a change in functions might occur at the beginning of each calculation time step based on the results of the preceding time calculation time

step. Therefore, the actual time the control remains on will be rounded up to the nearest integer multiple of the time step that meets or exceeds the specified minimum time on

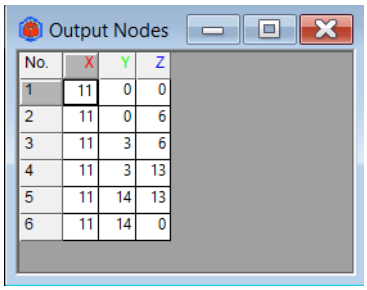
- Minimum time off: minimum duration that the control must continuously remain switched off. The actual time the control remains off will be rounded up to the nearest integer multiple of the time step that meets or exceeds the specified minimum time off.
- Function from: the function reference (E##, H##, I##, M##, P##, R##, S##, T## or V##, with ## = 01 up to 99) to a function defined in the **Functions window** and referred to in the **Colours window**, **Node BCs window**, **Borderface BCs window**, **Material BCs window** or **Ventilation Flows window**.
  The "function from" is applied when the control is switched off.
- Function to: function reference (E##, H##, I##, M##, P##, R##, S##, T## or V##, with ## = 01 up to 99) to a function defined in the **Functions window**, which replaces the "function from" while the control is switched on.
  The "function from" and "function to" must both be of the same type (i.e. both function references must begin with the same letter).
- Test $\Delta\theta$: switch to test the temperature difference between two zones (of a ventilation path), as (additional) condition to switch on a ventilation control (only available if "function from" and "function to" start with the letter V).
  For heating: The zone 1 temperature ($\theta 1$) must exceed the zone 2 temperature ($\theta 2$) by at least the minimum required temperature difference for control to be enabled.
  For cooling: The zone 1 temperature ($\theta 1$) must be lower than the zone 2 temperature ($\theta 2$) by at least the minimum required temperature difference for control to be enabled.
- $\Delta\theta$: minimum required temperature difference [°C] between two zones (of a ventilation path).
- Zone $\theta 1$ for temperature difference test, which is defined by its colour number (between 0 and 255) in the **Colours window**. The colour must be defined in the ventilation path.
- Zone $\theta 2$ for temperature difference test, which is defined by its colour number (between 0 and 255) in the **Colours window**. The colour must be defined in the ventilation path.

B.21 Output nodes

Output nodes are points to which reference is made in a report text output (cf. section F.3.3) to show the course of temperatures in those points as a function of time.
Output nodes can also be used as sensor points in controls (cf. section B.20).

Output nodes are defined in the **Output Nodes window**. The definition of an output node consists of a sequence number and the point coordinates (X,Y,Z or R,C,L depending on the active coordinate system). The position of the current output node is represented in the **Image window** by the intersection of three lines parallel to the coordinate axes (coloured red, green and blue).

Output nodes can be inserted (using *Edit → Insert Before* or *Edit → Insert After*) and deleted (using *Edit → Delete*).

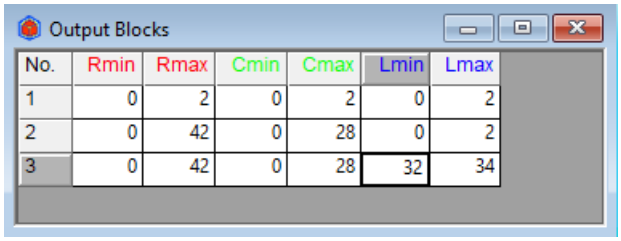


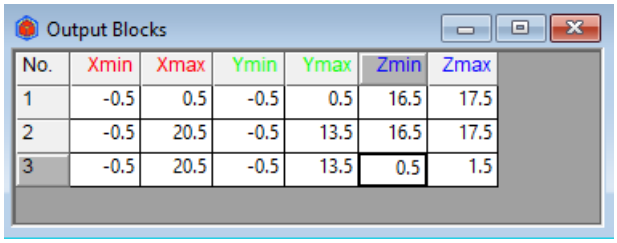
B.22 Output blocks

Output blocks can be used in a report text output (cf. section F.3.3) to show the course of incoming heat flows (in W) through conduction through the boundary faces of those output blocks. The heat flows by radiation or convection are not taken into account. So only the intersection parts of the boundary faces of an output block with the object material is important.

Output blocks are defined in the **Output Blocks window**.

A new output block can be added using *Edit → Insert Before* or *Edit → Insert After*. The definition of an output block consists of a sequence number and 6 coordinates.





The boundary faces of an output block always lie in between the grid coordinates as defined in the **Grid window**. In this way the boundary faces can be broken down in elementary boundary faces of control volumes around material nodes, through which the energy flows through conduction can easily be calculated.

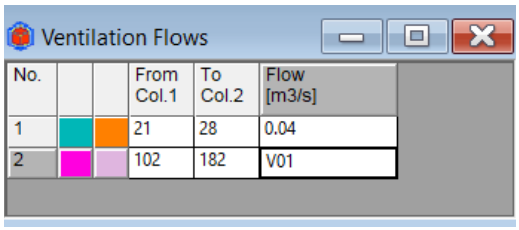
In the RCL coordinate system the coordinates of output blocks are represented by even numbers (0,2,4,...).

In the XYZ coordinate system an output block coordinate ends with .5 to indicate that its position is halfway between two consecutive grid locations (as defined in the **Grid window**).

The outline of the current output block is shown in the **Image window**.
Note that object clipping (C.10) has no effect on the calculation of heat flows into the output blocks; it has only a visual effect in the **Image window**.

B.23 Ventilation flows window

Ventilation flow paths can be defined in the **Ventilation Flows window**, opened with *Edit → Ventilation Flows*. Add new ventilation flows with the command *Edit → Insert After*.



For each ventilation flow definition line a ventilation source and ventilation destination must be entered. Both are identified by colour numbers to the corresponding boundary conditions.

The ventilation source (in the column with header "From Col. 1") can be any type of boundary condition (BC_SIMPL, BC_FRE_S, BC_FREE, BC_SKY or BC_NOSKY). The ventilation destination (in the column with header "To Col. 2") must be a boundary condition of type BC_FRE_S or BC_FREE (with unknown air temperature influenced by the incoming ventilation flow). Ventilation flows with destination a zone with fixed air temperature can not be entered, because they have no effect on the system.

The ventilation flow (last column) is a constant value or a reference to a ventilation function (V##) as defined in the **Functions window**.

The user should take care that all defined ventilation flows refer to a realistic ventilation scheme. A ventilation path can originate from an outside environment, pass through some internal zones and leave to another outside environment. But a ventilation path can also pass through a closed circuit of only internal zones.

All air cavities of type BC_FRE_S and BC_FREE that occur as ventilation source or ventilation destination in the **Ventilation Flows window** are automatically unlinked without the need of explicit definition as explained in section D.7.

The following error messages may occur when doing a calculation:

- "Syntax error in ventilation flow N":
  The ventilation flow in the **Ventilation Flows window** at line N contains a syntax error. Either the ventilation source is not a boundary condition (i.e. a colour of type BC_SIMPL, BC_FRE_S, BC_FREE, BC_SKY or BC_NOSKY), or the ventilation destination is not a boundary condition of type BC_FRE_S or BC_FREE, or the specific heat (c) and/or the density (ρ) of the ventilation source and destination are not equal (see B.10).
- "Error in ventilation system":
  The combination of all ventilation flows as defined in the **Ventilation Flows window** does not correspond to a physically correct ventilation system. This error occurs when the ventilation system has no unique solution (undetermined system).

B.24 Report Definition window

The contents of a report in the **Text Output window** (see Chapter F) after a calculation has been executed, is determined by the report definition and the report frequency (cf. section B.25).

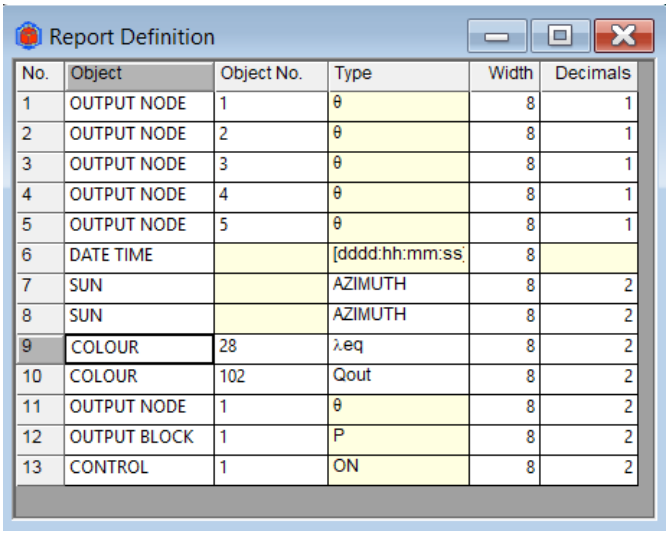
The input of the report definition is done in the **Report Definition window**, opened with *Edit → Report Definition*.

The report definition consists of several report items, one per line in the **Report Definition window**.

A new report item is added via *Edit → Insert Before* or *Edit → Insert After*.

During the generation of a report in the **Text Output window**, a report line is written per report step (which equals the calculation time step or a multiple of it, as defined by the report frequency).

The report line consists of the actual values of the report items (separated by tabs) at the current time step.



No.	Object	Object No.	Type	Width	Decimals
1	OUTPUT NODE	1	θ	8	1
2	OUTPUT NODE	2	θ	8	1
3	OUTPUT NODE	3	θ	8	1
4	OUTPUT NODE	4	θ	8	1
5	OUTPUT NODE	5	θ	8	1
6	DATE TIME		[dddd:hh:mm:ss	8	
7	SUN		AZIMUTH	8	2
8	SUN		AZIMUTH	8	2
9	COLOUR	28	λeq	8	2
10	COLOUR	102	Qout	8	2
11	OUTPUT NODE	1	θ	8	2
12	OUTPUT BLOCK	1	P	8	2
13	CONTROL	1	ON	8	2

A report item is defined by the following elements (corresponding to the different columns):

- Object: a toggle with values TIME, DATE TIME, COLOUR, 2 COLOURS, NODE BC, BORDER BC, MATERIAL BC, OUTPUT NODE, OUTPUT BLOCK, CONTROL, SUN.
- Object No., which meaning is dependent on the selected object:
  - TIME: not applicable.
  - DATE TIME: not applicable.
  - COLOUR: colour number (cf. first column in **Colours window**).
  - 2 COLOURS: 2 colour numbers (cf. first column in **Colours window**).
  - NODE BC: node boundary condition number (cf. first column in **Node BCs window**)
  - BORDER BC: borderface boundary condition number (cf. first column in **Borderface BCs window**).
  - MATERIAL BC: material boundary condition number (cf. first column in **Material BCs window**).
  - OUTPUT NODE: output node number (cf. first column in **Output Nodes window**).
  - OUTPUT BLOCK: output block number (cf. first column in Output **Blocks window**).
  - CONTROL: control number (cf. first column in **Controls window**).

- SUN: not applicable.

- Type, a toggle with values dependent on the selected object:

Object	Type	Meaning
TIME	[s]	time in seconds of the current time step
DATE TIME	[dddd:hh:mm:ss]	time in days, hours, minutes and seconds of the current time step
COLOUR	θ_{min}	minimum temperature [°C] (for colour of type MATERIAL or EQUIMAT)
	θ_{max}	maximum temperature [°C] (for colour of type MATERIAL or EQUIMAT)
	$\Delta\theta_{max}$	maximum temperature difference (θ_{max} - θ_{min}) [°C] (for colour of type MATERIAL or EQUIMAT)
	θ_{mean}	mean temperature ((θ_{min} + θ_{max})/2) [°C] (for colour of type MATERIAL or EQUIMAT)
	Q_{in}	heat flow into object out of boundary condition [W] (for colour of type BC_SIMPL, BC_FRE_S, BC_FREE, BC_SKY, BC_NOSKY or TRANSMAT)
	Q_{out}	heat flow out of object into boundary condition [W] (for colour of type BC_SIMPL, BC_FRE_S, BC_FREE, BC_SKY, BC_NOSKY or TRANSMAT)
	λ	(equivalent) thermal conductivity [W/(m.K)] (for colour of type MATERIAL, EQUIMAT or TRANSMAT)
	θ	boundary condition temperature [°C] (for colour of type BC_SIMPL or BC_FRE_S)
	h	total surface heat transfer coefficient [W/(m^2.K)] (for colour of type BC_SIMPL or BC_FRE_S)
	q	heat flux into boundary condition [W/m^2] (for colour of type BC_SIMPL, BC_FRE_S, BC_FREE, BC_SKY or BC_NOSKY)
	θ_a	air temperature [°C] (for colour of type BC_FREE, BC_SKY or BC_NOSKY)
	h_c	convective heat transfer coefficient [W/(m^2.K)] (for colour of type BC_FREE, BC_SKY or BC_NOSKY)
	P_c	convective power into boundary condition [W/m] (for colour of type BC_FREE or BC_FRE_S)
	θ_r	radiation temperature [°C] (for colour of type BC_SKY)
	q_{sol}	(area weighted average) absorbed solar flux on all faces oriented to Xmin/Xmax/Ymin/Ymax/Zmin/Zmax (including faces that do not receive any solar radiation) [W/m^2] (for colour of type MATERIAL or EQUIMAT)
	E_{tot}	Total solar irradiance in interface with solar zone [W/m^2]

Object	Type	Meaning
	E_{dir}	Direct solar irradiance in interface with solar zone [W/m^2]
	E_{dif}	Diffuse solar irradiance in interface with solar zone [W/m^2]
	E_{gr}	Ground reflected solar irradiance in interface with solar zone [W/m^2]
2 COLOURS	θ_{min}	minimum temperature [°C] occurring in set of 2 colours
	θ_{max}	maximum temperature [°C] occurring in set of 2 colours
	$\Delta\theta_{max}$	maximum temperature difference (θ_{max} - θ_{min}) [°C] occurring in set of 2 colours
	θ_{mean}	mean temperature ((θ_{min} + θ_{max})/2) [°C] of set of 2 colours
NODE BC	θ	temperature [°C]
	P	power [W]
BORDER BC	θ	temperature [°C]
	q	heat flux [W/m^2]
MATERIAL BC	θ	temperature [°C]
	p	Power density [W/m^3]
OUTPUT NODE	θ	temperature [°C]
OUTPUT BLOCK	Q	heat flow through conduction into output block [W]
CONTROL	ON	control on/off status (0=off, 1=on)
SUN	AZIMUTH	sun azimuth [°] (0:south, 90:west, –90:east)
	ALTITUDE	sun altitude [°] (0:horizon, 90:zenith)

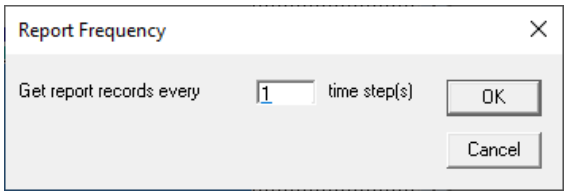
- Width: (maximum) number of characters for formatting the output values in the output report
- Decimals: number of digits after the decimal point, to which the output values are rounded.

If a report item definition is logically impossible, no values are given in the report (-).

B.25 Report Frequency

A report line in the report output is written for every time step or every fixed number of time steps. This number is set, before the calculation is started, in the **Report Frequency dialog box**, opened with *Calc → Report Frequency...*.

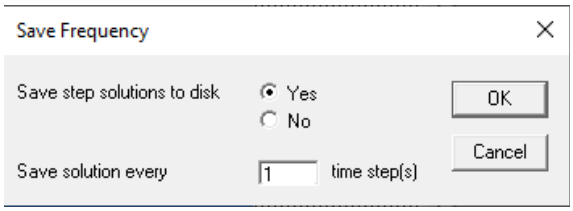
When a calculation is started, the amount of memory is allocated to save the report records during the calculation. At the end of the calculation the report records are saved in a text file and a CSV file (with same name and file folder as input data file, and extensions .txt and .csv). This report file can be viewed in the text output.



B.26 Save Frequency

During a calculation, time step solutions (i.e. the calculated temperatures in all calculation nodes and the absorbed solar fluxes) at every fixed number of time steps can be saved to disk to use later in graphic output or text output (to view the solutions at those time steps).

The save frequency must be set, before the calculation is started, in the **Save Frequency dialog box**, opened with *Calc → Save Frequency....*



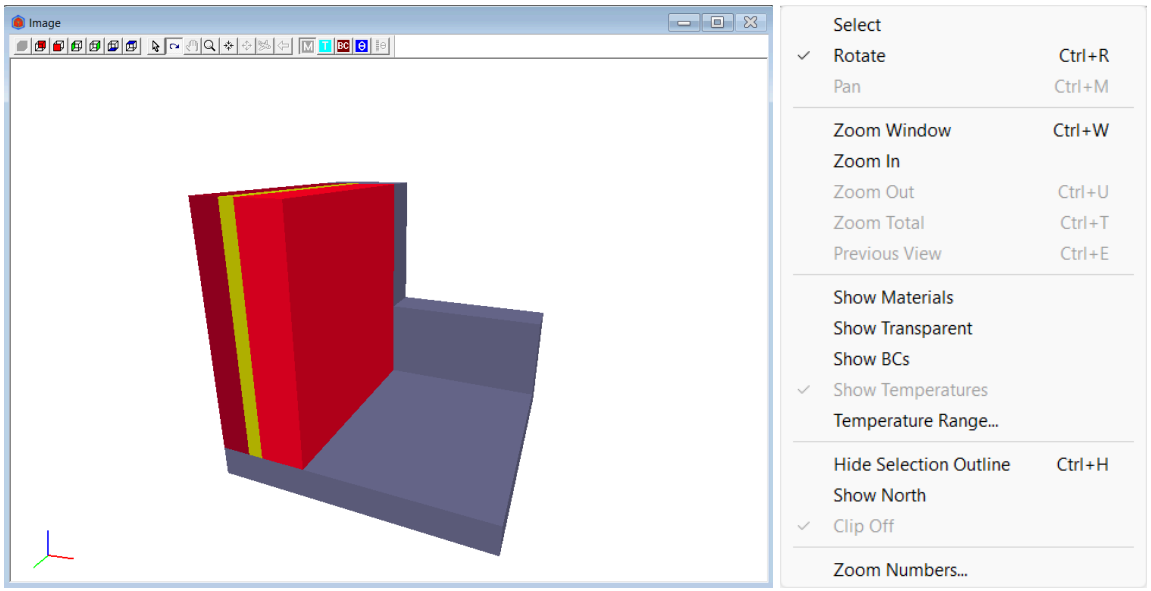
The node temperature solutions are saved in .sol files, .flx files and .sfx files with the same file name of the input data file appended with a 4-digit serial number. For example if the input data file is PilotexVoltra.vtr, then the solution files are PilotexVoltra0000.sol, PilotexVoltra0001.sol, PilotexVoltra0002.sol etc.

The format of the .sol and .flx files is identical to that of the TRISCO solution files.

CHAPTER C VIEWING THE MODEL

C.1 Image submenu

The object model as defined in the **Blocks window** and **Colours window** is shown in the **Image window**. The view of the object in the **Image window** can be altered using image functions. In the bottom left corner of the **Image window**, the 3 main coordinate axes are shown for reference (red for X or R, green for Y or C, and blue for Z or L).



The active view mode (*Rotate, Pan, Zoom Window* or *Zoom In*) and active object appearance (*Show Materials, Show Transparent, Show BCs, Show Temperatures* or *Show Single Temperature*) are checked in the Image submenu (see check mark in front of menu commands).

C.2 Viewpoint

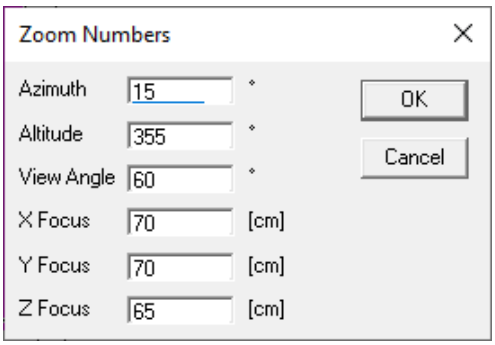
The viewpoint can be rotated or panned (i.e. moved laterally).

- *Image → Rotate*
  Enters rotate mode. Mouse dragging or pressing arrow keys (left, right, up, down, with **Image window** active) rotates the relative position of the viewpoint with respect to the object centre.
  The Z axis of the XYZ coordinate system should point upwards to the sky (or the layer axis of the RCL system should point downwards) to get a good behaviour of the rotate function.
  Rotate mode is activated automatically by holding down the mouse wheel when the **Image window** perspective (3D) projection is active (C.4).
- *Image → Pan*
  Enters pan mode (only possible when the current view is already zoomed in). Mouse dragging or pressing arrow keys (with **Image window** active) moves the relative position of viewpoint and object centre parallel to the projection plane.
  Pan mode is activated automatically by holding down the mouse wheel when the **Image window** 2D orthogonal projection is active (C.4).

The command *Settings → Drag Mode* allows to define the relative movement of viewpoint and object (as either "drag viewpoint" or "drag object", cf. section A.9).

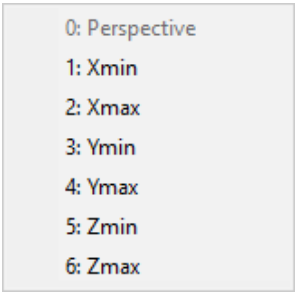
C.3 Zoom commands

- *Image → Zoom Window*
  Enters "zoom window" mode. This allows to draw a zoom rectangle using the mouse in the **Image window**, which defines the borders of a zoomed view.
  Press Esc to cancel the current zoom rectangle (before releasing the left mouse button).
- *Image → Zoom In*
  Enters "zoom in" mode. A following click using the left mouse button in the **Image window** defines the centre point of the next zoomed view. The zoom factor can be defined with *Settings → Zoom Factor...* (see A.9).
  Zooming in is also possible using forward scrolling with the mouse wheel (when the **Image window** is active).
- *Image → Zoom Out*
  Zooms out. The zoom factor can be defined with *Settings → Zoom Factor...* (see A.9).
  Zooming out is also possible using backward scrolling with the mouse wheel (when the **Image window** is active).
- *Image → Zoom Total*
  Shows total object.
- *Image → Previous View*
  Shows the previous zoom view. (More than one previous view cannot be recalled.)
- *Image → Zoom Numbers...*
  Opens a dialog box with the current zoom parameters.
  Enter new parameters to change the view. Use preferably rounded numbers to facilitate restoring the same view at a later moment.



C.4 Projection

Projection submenu



The projection of the model in the **Image window** can be toggled between

- *Projection → Perspective*: perspective view of the 3D model.
- *Projection → Xmin...:* 2D orthogonal view of the model onto the YZ plane with minimum X coordinates, looking in the positive X direction.
- *Projection → Xmax...:* 2D orthogonal view of the model onto the YZ plane with maximum X coordinates, looking in the negative X direction.
- *Projection → Ymin...:* 2D orthogonal view of the model onto the XZ plane with minimum Y coordinates, looking in the positive Y direction.
- *Projection → Ymax...:* 2D orthogonal view of the model onto the XZ plane with maximum Y coordinates, looking in the negative Y direction.
- *Projection → Zmin...:* 2D orthogonal view of the model onto the XY plane with minimum Z coordinates, looking in the positive Z direction.
- *Projection → Zmax...:* 2D orthogonal view of the model onto the XY plane with maximum Z coordinates, looking in the negative Z direction.

C.5 Object appearance

Object compilation

The object appearing in the **Image window** is the result of processing the blocks list in the **Blocks window**.

Each block has a colour. The colour (defined in the **Colours window**) refers to either a material (type MATERIAL, EQUIMAT or TRANSMAT) or a boundary condition (type BC_SIMPL, BC_FRE_S, BC_FREE, BC_SKY or BC_NOSKY).

A *material block* (which refers to a material colour) is added to the resulted object.

A *boundary condition block* (which refers to a boundary condition block) is subtracted from the resulted object. The boundary condition is active on all object surfaces adjacent to the boundary condition block.

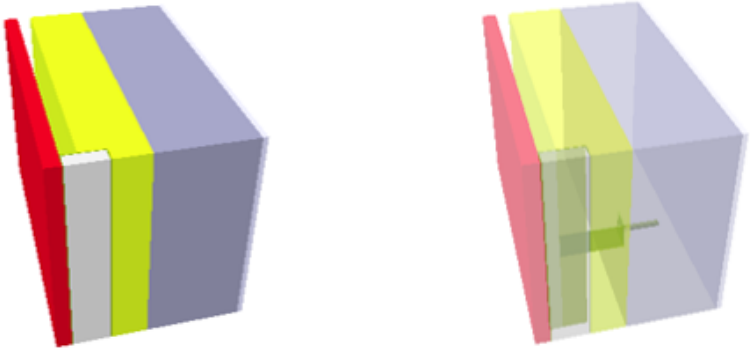
The *superposition rule* in the **Blocks window** makes that blocks with higher sequence number overwrite the common parts of blocks with lower sequence number.

View options

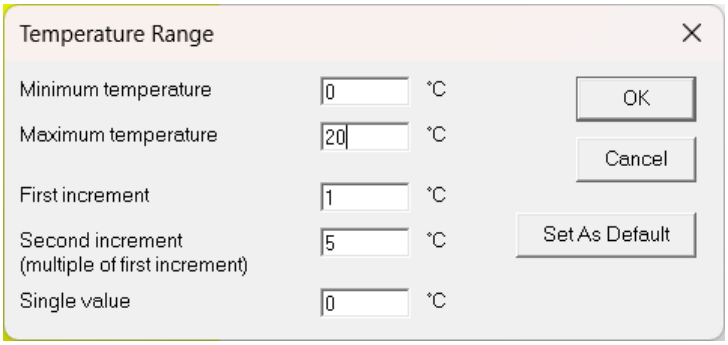
The colour of the surfaces of the material blocks of the object in the **Image window** can be modified:

- *Image → Show Materials*
  Shows the material colours of the material blocks in the object.

- *Image → Show Transparent*
  Shows the object with transparent materials, so that the inside of the object can be viewed.



- *Image → Show BCs*
  Shows the colours of the boundary conditions which are active on the material blocks in the object.
  When there are no explicit boundary conditions active on a material block surface, then an adiabatic boundary condition (i.e. without any heat transfer) is supposed. This implicit adiabatic boundary condition is coloured light yellow on the material blocks.
- *Image → Show Temperatures*
  Shows the calculated temperatures on the material blocks in the object.
  This option is only available if a calculation solution is present.
  The thermal palette making up the colours of the temperature scale are interpolated from values defined in a .rgb file (see A.9). This is a text file containing red, green and blue components (values from 0 up to 255) for increasing temperature values.
  The legend of used colours can only be made visible in the **Graphics Output window** (cf. section E.1).
  The temperature scale is defined in *Image → Temperature Range....*
- *Image → Temperature Range...*
  Opens a dialog box to define the parameters of a temperature view.
  The first increment is used as step value in *Image → Temperature Range....*
  (The second increment is only used with isothermal lines in the **Graphics Output window**)
  This command is only enabled when the menu option "Show Temperatures" is checked.



Remark

Before starting a calculation, it is advised to check carefully the surface boundary conditions active on the object surface using the command *Image → Show BCs.*

C.6 Quick block selection using the mouse

It is possible to select a block in the **Blocks window** from the **Image window** using the mouse.

The current object appearance mode should be:

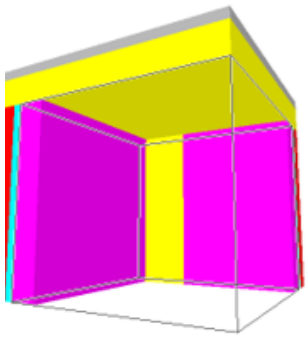
- "Show Materials" (activated through *Image* → *Show Materials*) to select a material block;
- "Show BCs" (*Image* → *Show BCs*) to select a boundary condition block.

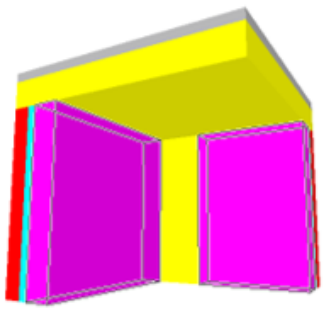
Point to the desired surface in the **Image window** and click (double click with left mouse button, or single click with right mouse button) to move to the underlying block in the **Blocks window**.

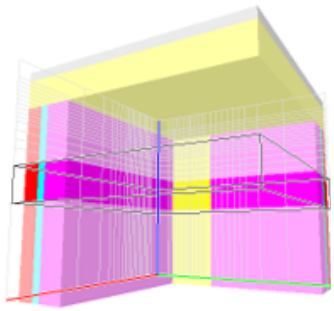
C.7 Active item feedback in the Image window

The active item of the current input window is highlighted in the **Image window**:

- From the **Blocks window**: edges of current block (in wireframe).
  Example: see figure below left.
- From the **Colours window**: object parts corresponding to the current colour (in wireframe).
  Example: see figure below centre.
  The volume of all blocks having the current colour is shown in the status bar when the active cell is upon the colour name.
- From the **Grid window**: edges of box surrounding the current mesh (in wireframe).
  Example: see figure below right.
  The colours of object surfaces not intersecting the grid mesh box are dimmed.
  The edges of the surrounding box of the entire grid space closest to the coordinate axes are coloured red (X or R), green (Y or C) and blue (Z or L).







Analogous feedback for the current item in the **Image window** is available from the other input windows.

The feedback lines can be removed by clicking the toggle *Image* → *Hide Outline*. Click this command again to show the feedback again.

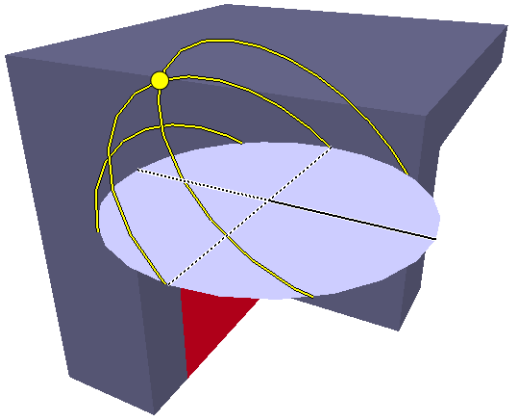
C.8 Show North

Menu command: *Image* → *Show North*

Above the object the horizontal plane is shown as a light blue disk (from the top side) or a dark blue disk (from the underside). The full line points to the North.

The sun paths are shown for winter solstice (21 December), vernal equinox (20 March) and summer solstice (21 June).

To hide the north orientation, call *Image → Show North* again.



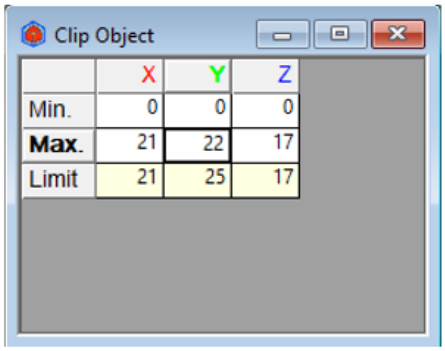
C.9 Pick temperature using the mouse

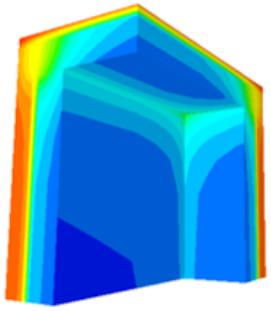
When the **Image window** is active (i.e. its title bar is blue) and the temperatures are shown (*Image → Show Temperatures*), then the surface temperature at the mouse cursor is displayed in the status bar (bottom left of the screen).

The number of decimals can be modified via the Decimals dialog box from the **Text Output window** (cf. section F.4.1).

C.10 Object clipping

Menu command: *Edit → Clip Object*





Part of the object can be shown in the **Image window** by clipping off all object parts outside a grid interval in any of the three coordinate directions. In this way cross-sections through all grid planes can be shown.

The clip borders are defined in the **Clip Object window** (opened with *Edit → Clip Object*). Editing the clip borders is immediately reflected in the **Image window**.

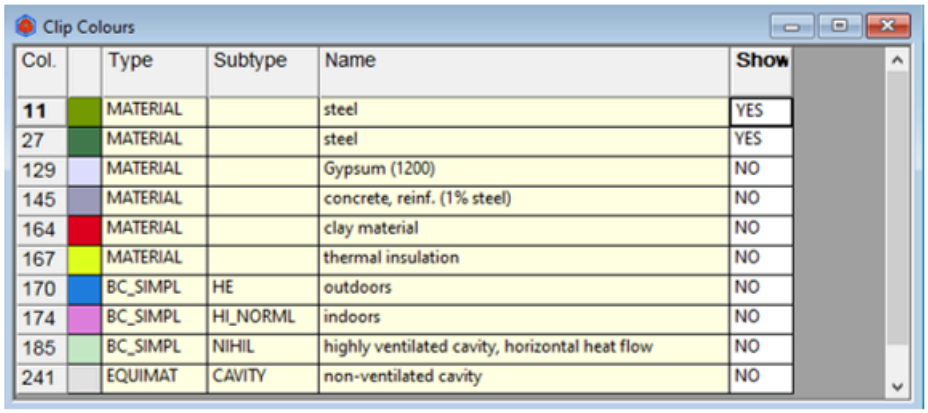
Clipping can be reset (to view the total object) with the command *Image → Clip Off*.

The object clip settings have no effect on the calculation. The calculation is always executed on the total object.

It is possible to reduce the object to the current clip settings with the command *Blocks → Reduce to Clip...* (cf. section B.9.7).

C.11 Colour clipping

Menu command: *Edit → Clip Colours*





Object parts of each colour can be clipped off in the **Image window**. The visibility status is defined in the **Clip Colours window** (opened with *Edit → Clip Colours*). This function can come in handy for example to visualise the temperatures in parts of the model which are otherwise hidden.

The colour clip settings are also applied in the **Graphic Output window**.

Colour clipping has no effect on the calculation. The calculation is always executed on the total object.

It is possible to reduce the object to the current clip settings with the command *Blocks → Reduce to Clip...* (cf. section B.9.7).

C.12 Coordinates window

Menu command: *Edit → Coordinates*

The **Coordinates window** shows the absolute coordinates for all grid planes.

The coordinate system is XYZ or RCL (see A.9).

The origin on each axis can be moved to any grid plane position via *Coord → Set Origin*. The origin is used as anchor point for coordinate transformations (*Blocks → Transform Coordinates...*: rotation, scaling) and file insertion (*Blocks → Insert File...*).

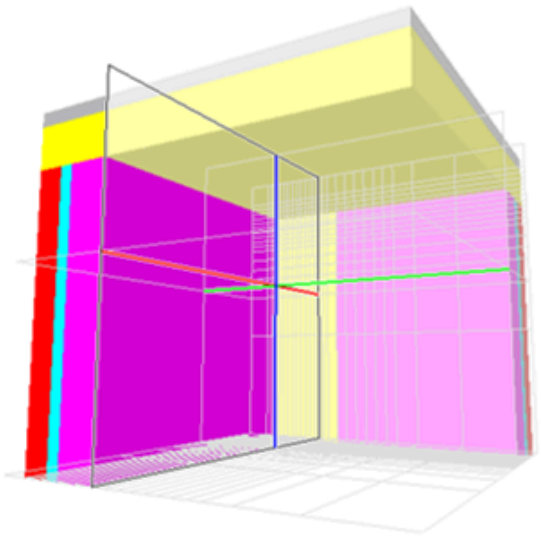
The coordinate values cannot be edited in the **Coordinates window**. Coordinates are modified via the **Grid window** (*Edit → Grid*).

The background colours of the coordinates modify according to the underlying minimum grid (cf. section B.7).

The position of the origin and coordinate axes (X or R in red, Y or C in green, Z or L in blue) are shown in the **Image window**. The position of the current coordinate is at the transition to the dimmed block colours (using *Image → Show Materials* or *Image → Show BCs*).

Coordinates

No.	X [cm]	Y [cm]	Z [cm]
0	-40.000	-40.000	-70.000
1	-30.000	-30.000	-20.000
2	-27.500	-27.500	0.000
3	-25.000	-25.000	10.000
4	-22.500	-22.500	15.000
5	-20.000	-20.000	20.000
6	-17.500	-17.500	25.000
7	-15.000	-15.000	27.500
8	-12.500	-12.500	30.000
9	-10.000	-10.000	32.500
10	-7.500	-7.500	35.000
11	-5.000	-5.000	37.500
12	-2.500	-2.500	40.000
13	0.000	0.000	42.500
14	2.500	2.500	45.000
15	5.000	5.000	47.500
16	10.000	10.000	50.000
17	15.000	15.000	60.000



CHAPTER D CALCULATION

D.1 Automatic calculation of thermal properties

In the **Colours window** colour types and subtypes can be selected which contain an automatic calculation of thermal properties in accordance with international standards (equivalent thermal conductivity or convective heat transfer coefficient).

During the input of new data the calculation is based on the default temperature values from the relevant standards, or on the default temperature difference across airspace as defined by the user in the **Calculation Parameters dialog box** (see D.2.4) if the calculated thermal property is not standardized.

During a system calculation the thermal properties are recalculated based on the real temperature values, obtained by the global thermal calculation. The non-linear calculation is split in several *iteration cycles*. Each iteration cycle solves a linear system. The linear system is based on updated thermal properties derived from the temperatures available from the solution of the previous iteration cycle.

D.1.1 Colour type EQUIMAT

Colour modelled as *equivalent material* (with *single equivalent thermal conductivity* λ_{eq} for combined convective and radiation heat transfer)

Type	*Subtype*	*Meaning*	*Standard*
EQUIMAT	NIHIL	Thermal conductivity λ defined by user	
	CAVITY	Unventilated (internal) air cavity with given heat flow direction	EN10077 EN6946
	CAVITY_E	Ventilated (external) air cavity with given heat flow direction	EN10077
	LAYER	Unventilated air layer with given heat flow direction	EN6946

Characteristic dimensions of an air cavity

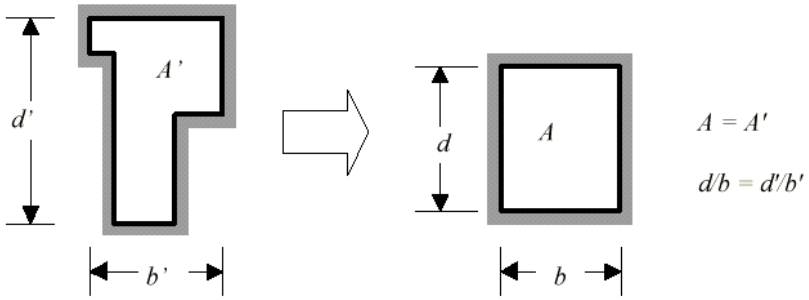
Consider a characteristic 2D section of the air cavity, containing the heat flow direction (parallel to a coordinate axis) and the smaller of the other two directions of the cavity. The characteristic dimensions in the 2D section are called *depth* (parallel to the heat flow direction) and *width* (orthogonal to the heat flow direction).

If the air cavity is not a rectangular parallelepiped (or cuboid) shape, the depth and width are calculated from an equivalent rectangular parallelepiped with the same aspect ratio:

d' = size of surrounding box of air cavity in depth direction
b' = size of surrounding box of air cavity in width direction
V' = volume of surrounding box of air cavity
V = real volume of air cavity
$d = d' \cdot s$
$b = b' \cdot s$

$$s = \sqrt{\frac{V}{V'}}$$

If the air cavity has a uniform length in the third dimension (perpendicular to depth and width), then the calculation corresponds to the 2D calculation rule in EN ISO 10077-2.



The calculated dimensions *d* and *b* are further used in the formulas below.

Equivalent thermal conductivity of unventilated air cavity (subtype CAVITY)

$\lambda_{eq} = (h_c + h_r) \cdot d$

- h_c = convective heat transfer coefficient (from warm surface to cold surface)
  if b < 0.005 m (EN ISO 10077-2), then

  $h_c = \frac{C_1}{d}$

  if b ≥ 0.005 m (EN ISO 10077-2) or for any b (EN ISO 6946), then

  $h_c = \max\left\{\frac{C_1}{d}, C_2 \cdot (\Delta\theta_{ss})^{C_3}\right\}$

  $\Delta\theta_{ss}$ = maximum surface temperature difference in air cavity ($\Delta\theta_{ss}$ ≥ 5°C for EN ISO 6946)
  or $\Delta\theta_{ss} = \Delta\theta_{ss,default}$

Standard	*Heat flow direction*	C_1	C_2	C_3	$\Delta\theta_{ss,default}$
EN ISO 10077-2	horizontal	0.025	0.73	0.333333	10°C
EN ISO 6946	horizontal	0.025	0.73	0.333333	5°C
	upward	0.025	1.14	0.333333	5°C
	downward	0.025	$0.09 \cdot d^{-0.44}$	0.187	5°C

- h_r = radiative heat transfer coefficient
  EN ISO 10077-2: $h_r = 4 \cdot 5.67 \cdot 10^{-8} \cdot (\theta_m + 273.16)^3 \cdot E \cdot F$

  EN ISO 6946: $h_r = \frac{4 \cdot 5.67 \cdot 10^{-8} \cdot (\theta_m + 273.16)^3}{\left(\frac{1}{E} + \frac{1}{F} - 1\right)}$

  $E = \frac{1}{\frac{1}{\varepsilon_1} + \frac{1}{\varepsilon_2} - 1}$

  $F = \frac{1 + \sqrt{1 + \left(\frac{d}{b}\right)^2} - \frac{d}{b}}{2}$

$$\theta_m = \frac{\theta_{s,min} + \theta_{s,max}}{2}$$

$\theta_{s,min}$ = minimum surface temperature in air cavity [°C]
$\theta_{s,max}$ = maximum surface temperature in air cavity [°C]
or θ_m = 10°C (default mean temperature, see D.2.4)
ε_1, ε_2 = emissivities at warm and cold sides of air cavity (default values: ε_1 = ε_2 = 0.9)

Equivalent thermal conductivity of slightly ventilated air cavity (subtype CAVITY_E, standard EN10077)

An air cavity is considered to be slightly ventilated when there is an open connection to an exterior or interior boundary condition with a slit between 2 and 10 mm, and $d \geq b$.
$\lambda_{eq} = 2 \cdot (\lambda_{eq}$ of unventilated air cavity).

Equivalent thermal conductivity of air layer (subtype LAYER)

Standard EN ISO 6946: in the formula above for subtype CAVITY, set *d/b* = 0 for h_r.

Adiabatic boundaries

An adiabatic boundary (i.e. a boundary through which there is no heat transfer) can be considered as a symmetry axis (with symmetric heat flow pattern).
When there is an adiabatic boundary at one side of the width direction, the equivalent width *b* is automatically doubled for subtype CAVITY.
When there are adiabatic boundaries at both sides of the width direction, the equivalent width *b* is automatically considered to be infinite for subtype CAVITY (so *d/b* = 0, as for subtype LAYER).

D.1.2 Colour type BC_SIMPL or BC_FRE_S

Purpose: simplified surface boundary condition (using a global surface heat transfer coefficient). For BC_SIMPL, the environment temperature is defined by the user. For BC_FRE_S, the environment temperature is unknown (free-floating).

Type	*Standard*	*Subtype*	*Physical heat flow*	*Meaning*
BC_SIMPL	NIHIL	NIHIL		Global surface heat transfer coefficient h and temperature θ defined by user
	EN10077	HI_NORML	horizontal	Normal h at interior surface: h = 7.7 W/(m^2.K), θ_i = 20°C
		HI_REDUC	horizontal	Reduced h at interior surface: h = 5 W/(m^2.K) , θ_i = 20°C
		HE		h at exterior surface: h = 25 W/(m^2.K), θ_e = 0°C
		HI_CAV_E	horizontal	Slightly ventilated air cavity: h = 3.33 W/(m^2.K), θ_i = 20°C
		HE_CAV_E	horizontal	Slightly ventilated air cavity: h = 3.33 W/(m^2.K), θ_e = 0°C
	EN6946	HI	horizontal	h at interior surface: h = 7.7 W/(m^2.K)
			upward	h at interior surface: h = 10 W/(m^2.K)
			downward	h at interior surface: h = 5.9 W/(m^2.K)
		HE		h at exterior surface: h = 25 W/(m^2.K)

D.1.3 Colour type BC_FREE

(This type requires the RADCON module, see Chapter I)

Purpose: enclosure with unknown (free-floating) air temperature (single thermal node) to be calculated from convective heat balance, and radiation based on view factors.

Type	*Subtype*	*Meaning*
BC_FREE	NIHIL	Thermal conductivity λ defined by user
	CONVEC	Large air space, without preferential heat flow direction*
	CAVITY	Unventilated (internal) air cavity with given heat flow direction

*typically larger than 0.3m

CAVITY :
The CAVITY subtype is to be used when a global heat flow direction is assumed (surface-to-surface heat transfer with an air temperature in between temperature of 'hot' and 'cold' surface temperatures, e.g. typically used for a cavity in a façade).

The characteristic dimensions of the air cavity, *d* (= depth, parallel to heat flow) and *b* (= width), are calculated as for colour type EQUIMAT (see D.1.1).

The convective surface heat transfer coefficient h_c is calculated as for colour type EQUIMAT (see D.1.1). The values calculated according to the different standards in D.1.1 are defined

however as surface-to-surface. Since the convective surface heat transfer coefficient h_c for BC_FREE is surface-to-air, the values calculated in D.1.1 are doubled.

h_c = convective surface heat transfer coefficient (surface-to-air)

$h_c = 2 \cdot (h_{c,\,EQUIMAT})$

CONVEC:

The subtype CONVEC must be used when no global heat flow direction can be assumed in the enclosure (surface-to-air heat transfer with an air temperature higher or lower than surface temperatures, e.g. typically used for a room).

h_c= convective surface heat transfer coefficient (surface to air)

$h_c = 2 \cdot C_2 \cdot (\Delta\theta_{sa})^{C_3}$

$\Delta\theta_{sa}$= greatest difference between air temperature and surface temperature

$\Delta\theta_{sa} = \max(|\theta_a - \theta_{s,min}|, |\theta_a - \theta_{s,max}|)$

θ_a = air temperature
$\theta_{s,min}$ = minimum surface temperature in contact with given boundary condition colour
$\theta_{s,max}$ = maximum surface temperature in contact with given boundary condition colour
or $\Delta\theta_{sa}$= calculation parameter "Default temperature difference for h_c calculation (subtype CONVEC)" (see D.2.4)

Heat flow direction	C_2	C_3
horizontal	0.73	0.333333
upward	1.14	0.333333
downward	0.09	0.187

D.1.4 Colour type BC_SKY or BC_NOSKY

(This type requires the RADCON module, see Chapter I)

Purpose: environment with known air temperature and either user defined radiation temperature (BC_SKY) and radiation based on view factors or no radiation exchange (BC_NOSKY).

Type	*Subtype*	*Meaning*
BC_SKY or BC_NOSKY	NIHIL	Convective surface heat transfer coefficient h_c defined by user
	CONVEC	Natural convection surface-air (calculation of h_c: cf. above BC_FREE - CONVEC, see D.1.3)

D.1.5 Colour type TRANSMAT

(This type requires the RADCON module, see Chapter I)

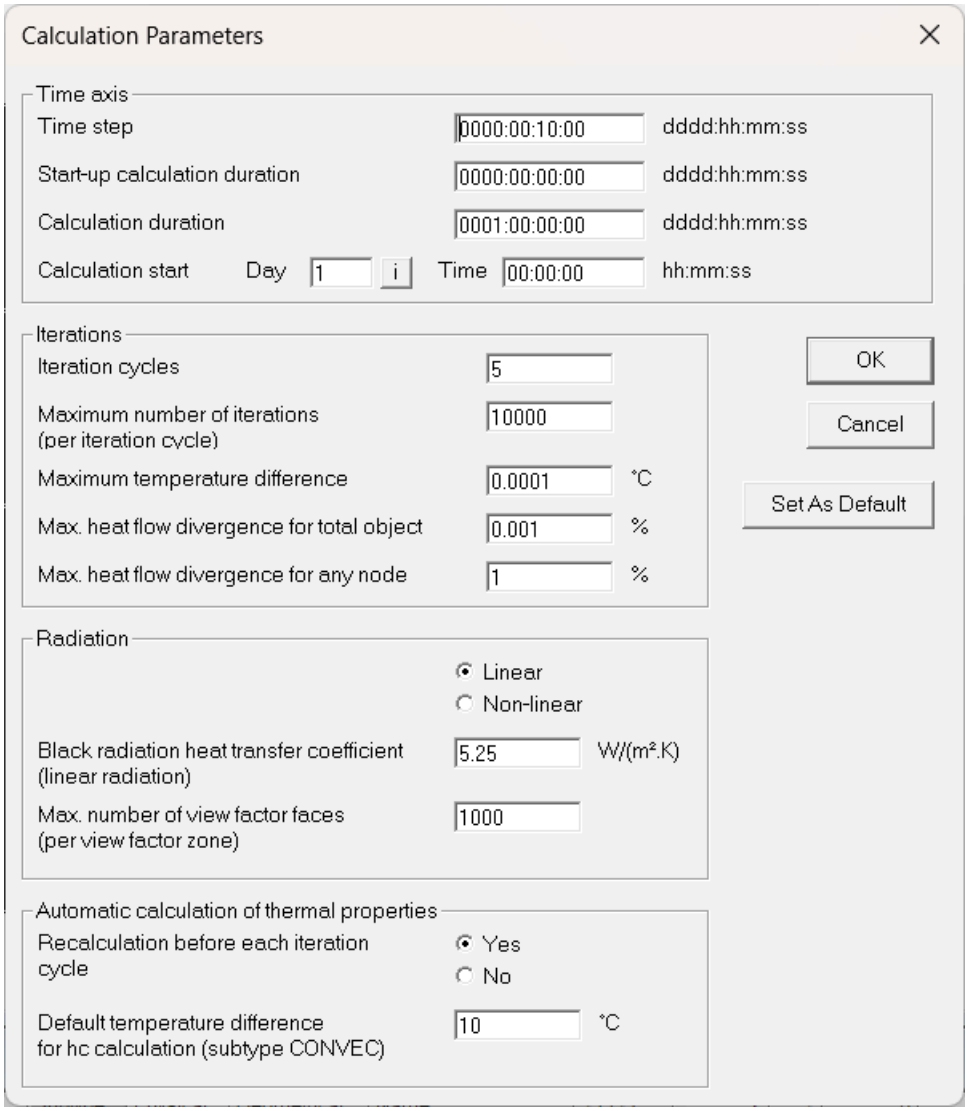
Purpose: air cavity modelled as *transparent material* (radiation based on view factors, cf. radiosity method (EN ISO 10077-2), and *equivalent thermal conductivity* λ_{eq} for convective heat transfer)

Type	*Subtype*	*Meaning*
TRANSMAT	NIHIL	Thermal conductivity λ defined by user
	CAVITY	Unventilated air cavity (calculation of λ_{eq}: cf. above EQUIMAT, CAVITY with h_r = 0, because radiation is calculated using view factors)

D.2 Calculation parameters

A system calculation (*Calc → Calc System*) is controlled by calculation parameters.

The menu command *Edit → Calc Parameters...* opens a dialog box in which the calculation parameters can be edited. The parameters not relevant to the current VOLTRA document are disabled (greyed in the dialog box).

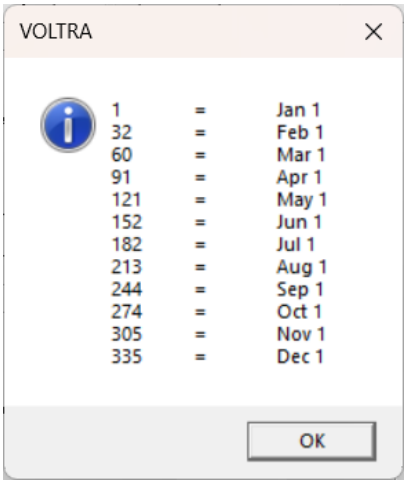


D.2.1 Time

- Time step: this is the time distance between two successive dynamic system calculations (time format: days, hours, minutes and seconds separated by colons).
- Start-up calculation duration: this is the time duration before the actual start of the calculation (where reporting starts) with the purpose to get good dynamic start values.

The boundary condition values of time-dependent functions are taken before the time origin of those functions, so that the actual calculation (at the end of the start-up calculation duration) starts at time 0. Internal functions which are not periodic (i.e. functions of type STEP0 and STEP1), get the function value at time 0 during the entire start-up calculation duration. Periodic functions are extended for negative time values with the same periodic function course. For external functions the "previous function" is taken, as defined in the function editor FUNCEDIT (cf. Chapter H).

- Calculation duration: duration of the dynamic calculation during which time step results and report items can be saved (e.g. to produce a graphic animation).
- Calculation start: definition of the day number (between 1 and 365) and exact time of day to indicate the moment in the year where the calculation starts. This is important when the used (external) function is related to a calendar date: e.g. a test reference year, with hourly climatic data of a typical year in a given location. The start date of the external function or weather data file may be different from the start date of the calculation.
  The button "i" near the input field gives, when pressed, the day numbers for the first day of each month for information.



In case of solar radiation functions (G## and D##) using an external data file with data formatted as a step function (B.12.2) or a weather file (B.12.3), the time step and calculation start have to be defined in such a way that the simulation hits the hours precisely. If this condition is not met, an error message is shown when starting the calculation.

D.2.2 Iterations

Iteration cycles

An iteration cycle is part of a system calculation (cf. section D.5) and involves building and solving a linear system of equations. A non-linear system is solved by stepwise linearisation. Each step using new linearised coefficients corresponds to an iteration cycle. This is needed for automatic refinement of calculated thermal properties (equivalent thermal conductivities and convective heat transfer coefficients) or non-linear infrared radiation. Before each new iteration cycle the linearised thermal resistances are recalculated based on the last available solution (i.e. from the previous iteration cycle).

The given number of iteration cycles is only applied for the first time step (i.e. solution of the initial steady-state system, cf. section D.6). If the initial temperature differences are rather high, the number of iteration cycles should also be high (to get reliable start values of

temperature dependent thermal properties). For the next time steps there is always only one single iteration cycle (independent of the given number of iteration cycles in the **Calculation Parameters dialog box**).

Iteration stopping criteria

The linear system of each iteration cycle is solved using an iterative method.
The iteration process is stopped when the *maximum number of iterations* (given as calculation parameter; by default 10000) is reached, or more likely when all three following conditions are fulfilled:

- *Temperature difference condition*
  For every system node the absolute value of the difference between the temperatures obtained in the current and previous iteration step must be smaller than the parameter *maximum temperature difference* (by default 0.0001°C).
- *Energy heat balance for the total object*
  The residual heat flow into or outside the object compared to the total heat flow passing through the object must be less than the parameter *maximum heat flow divergence for total object* (by default 0.001 %).
- *Worst energy heat balance for all control volumes*
  For all system nodes the residual heat flow into or outside the control volume compared to total heat flow passing through the control volume must be less than the parameter *maximum heat flow divergence for any node* (by default 1 %).

Definition of heat flow divergences: see section D.10.

D.2.3 Radiation

(see Chapter I for more on view factor based radiation heat transfer)

Linear radiation

The linear radiation between two black surfaces is:

$$q_r = F_{ij} \cdot h_{rb} \cdot (\theta_i - \theta_j)$$

q_r = radiation heat flux [W/m²]
F_{ij} = view factor from surface i to surface j (= fraction of radiation leaving surface i that is intercepted by surface j)
h_{rb} = black radiation heat transfer coefficient [W/(m².K)]
θ_i = temperature of surface i [°C]
θ_j = temperature of surface j [°C]
The linear radiation between grey surfaces, having an emissivity (as defined in the **Colours window**), is derived from the radiation between the black surfaces by adding supplementary resistances between the grey and black surface nodes (cf. section I.2).

The *black radiation heat transfer coefficient* is a calculation parameter. The default value is 5.1 W/(m².K) (as in EN 673).

Non-linear radiation

When non-linear radiation is wanted, the coefficient h_{rb} is locally adjusted in function of the surface temperatures, before each new iteration cycle. The surface temperatures are known from the previous iteration cycle, or else from the previous existing solution file (when a

calculation is restarted). For the first iteration cycle when no solution file is present, the calculation parameter "black radiation heat transfer coefficient" is used.

The non-linear radiation model is physically more precise than the linear model (cf. section I.2).

Calculation of view factors

The view factors calculation between elementary surfaces is affected by the parameter *Maximum number of view factor faces (per view factor zone)*: radiative surfaces are clustered into larger view factors if the number of surfaces in a view factor zone exceeds this parameter (see D.3).

D.2.4 Automatic calculation of thermal properties

(See section D.1 for an overview of all automatically calculated thermal properties)

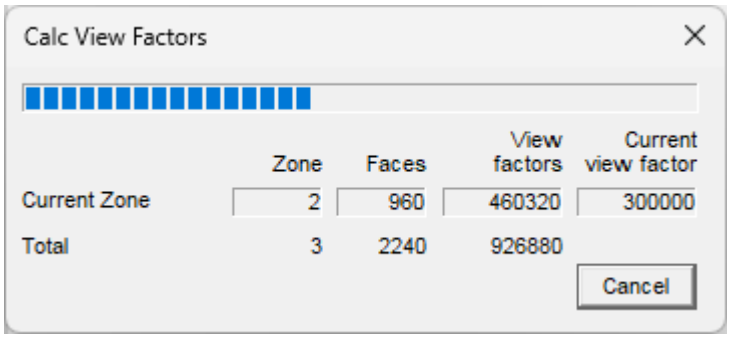
- *Recalculate before each iteration cycle*: the automatically calculated thermal properties (equivalent thermal conductivities and convective heat transfer coefficients) are recalculated before each iteration cycle (see D.2.2).
- *Default temperature difference across airspace*: parameter used in automatic calculation of thermal properties when no temperature solution and no default temperature difference according to the relevant standard are available (see D.1).

D.3 Calculation of view factors

Menu command: *Calc → Calc View Factors*

(This requires the RADCON module, see Chapter I)

A monitor shows the progress of the calculation.



The view factor boundary condition blocks (of colour type BC_FREE, BC_SKY, BC_NOSKY, TRANSMAT) are classified in separate zones. Adjacent boundary condition blocks of different colours must have the same type and are combined in the same view factor zone.

The same colour (of a view factor BC) is not allowed to occur in different zones (otherwise an error message appears when starting the calculation).

In each zone the number of elementary surfaces (defined by the grid meshes in the **Grid window**) is determined. The number of view factors in that zone is proportional to the square of the number of faces.

The geometric view factors between two (elementary) surfaces are calculated using analytic formulas.

The visibility between the surfaces (i.e. possible obstruction by other surfaces) is checked by a number of test rays between random points on both surfaces.

The calculated unobstructed view factors are reduced proportional to the number of obstructed test rays.

Afterwards the view factors are rebalanced so that the geometric laws are satisfied:

$$\sum_i F_{ij} = 1$$

$$A_i F_{ij} = A_j F_{ji}$$

The view factors are saved in a (text) file with the same name as the input data file, extension .vwf, and in the same file folder as the input data file.

View factor grid and system grid

In general, the *view factor grid* may be less fine than the *system grid* without loss of accuracy. Very small grid meshes do not bring significantly better results when the surfaces are at a relatively large distance from one another, although the calculation time is much larger.

Defining a maximum number of view factors per view factor zone clusters radiative surfaces (see D.2.3), resulting automatically in a view factor grid that is less refined than (but compatible with) the system grid. The view factor grid automatically created by this clustering is thus always a subset of the system grid. Conversely, the minimum grid (B.6.4) is always a subset of the view factor grid, thus imposing a limit to the clustering.

Alternatively, the user can define a grid manually specifically for the view factor calculation which is less refined than the grid for the system calculation (*Calc → Calc System*). It is however very important that the system grid is a refinement of the view factor grid, i.e. all grid coordinates of the view factor grid must also be present in the system grid.

The required view factors in the system calculation (using the system grid) are interpolated from the view factors obtained from the view factor calculation (using the view factor grid). The view factors (from the view factor calculation) are read from the file with extension .vwf[6] and with the same name as the input data file (used for the system calculation). This means that the same file name is used for both calculations:

- First the view factor calculation with a rough grid (results stored in .vwf file).
- Then grid refinement as required for the system calculation.
  (Of course this step is not obligatory, when the user accepts that view factor grid and system grid are equal.)
- Finally the system calculation (using the fine grid and view factor interpolation).

D.4 Calculation of solar fluxes

Menu command: *Calc → Calc Solar Fluxes*

This command is enabled when a solar zone is present in the input data. This is a colour of type BC_SIMPL or BC_SKY with sun toggle set to YES (B.10.8).

[6] Note that the syntax of the created view factor file (with extension .vwf) from VOLTRA v6.0w is modified compared to the format of the .vwf file of previous versions of VOLTRA.

Before executing this command be sure to define a text report (cf. section B.24), the report frequency (cf. section B.25) and/or the save frequency (cf. section B.26), otherwise no results of calculated solar fluxes are saved for text output (F.3.3) and/or graphic output (E.6.2).

First the view factors are calculated, if not yet present (see D.3).

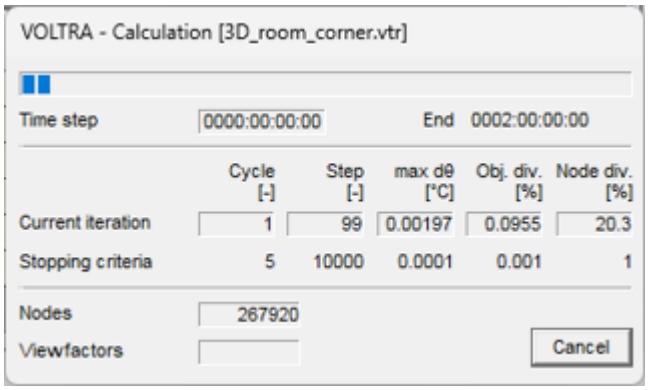
The solar fluxes are calculated upon all surfaces exposed to the sun. This includes direct solar radiation as well as diffuse solar radiation. Also (direct and diffuse) solar radiation through transparent walls is calculated. See D.9 for more details on solar radiation calculations.

The absorbed solar fluxes on each elementary surface are saved per time step in a file with extension .sfx. Global header information is saved at the start of the calculation in a file with extension .sfd. These files are read from the **Text Output window** and **Graphic Output window** when needed.

D.5 System calculation

After the input of data is finished, a calculation can be started with the command *Calc → Calc System*. First the view factors are calculated (if required, as in TRISCO), then the dynamic system calculation is started. During the system calculation a monitor is shown containing the current time step, the iteration parameters for the current time step, the number of system nodes, triangles and view factors.

After completion of each time step the **Image window** is updated showing the temperatures of the last time step.



If a start-up calculation is requested, first the start-up time steps are shown (from time 0000:00:00:00 up to the total start-up duration). Neither reporting nor solution saving is done during the start-up duration. After finishing the start-up calculation, the normal calculation duration begins. The time steps of the normal calculation duration are shown again starting from time 0000:00:00:00. During the normal calculation, reporting and solution saving are possible.

At the end of the calculation a report file is written, when at least 1 report item is defined in the **Report Definition window** (cf. sections B.24 and B.25). The content is the same as in the Report output (cf. section F.3.3). The report file has the same file name as the data file, and has the extension .txt.

A calculation can be interrupted at any time by pressing the Cancel button. If the "Save step solutions to disk" option is enabled in the **Save Frequency dialog box** (cf. section B.26), the solution files up to the last calculated time step can be viewed in the **Graphics Output**

window and **Text Output window**. If a report definition is given (cf. section B.24), the report records up to the last calculated time step can be viewed in the **Text Output window**.

If a calculation is restarted after it has been interrupted, the calculation restarts from the very beginning without reuse of the already calculated time step solutions.

It is possible to start from a predefined solution file StartVoltra.sol (cf. next section), to get an initial temperature distribution.

D.6 Calculation principles

For the first time step (of the start-up calculation duration if requested, else of the normal calculation duration) a steady-state system (as in TRISCO) based on the boundary condition function values evaluated at the initial time is set up and solved. Multiple iteration cycles (as defined as calculation parameter) are used to get good initial temperature dependent properties. If the initial temperature differences of the steady-state system are high, the number of iteration cycles should also be high (so that the temperature dependent thermal properties converge to stable values).

If a start solution (defined as .sol file) should be adopted at the first time step (of the start-up calculation duration if requested, else of the normal calculation duration), this solution should be renamed to the fixed file name StartVoltra.sol in the directory of the .vtr input data file.
The start solution is only used if this solution is compatible with the input data (i.e. same number of nodes, same number of object nodes, same number of blocks, same number of node boundary conditions).
Delete the file StartVoltra.sol after the calculation, otherwise it will be reused as start solution (if compatible with the input data) in all following calculations.

For the next time steps the boundary condition function values are updated and the material volume capacity is taken into account. The Cranck-Nicolson finite difference method is used to formulate the energy balance for all control volumes around material nodes. For all other nodes without capacity (e.g. black surface nodes, radiation and convection nodes of a boundary condition) the energy balance at the current time step is formulated.
Only 1 iteration cycle per next time step is considered, even if the calculation parameter "number of iteration cycles" has a higher value. This supposes fluent boundary condition changes (no abrupt discontinuities) during the calculation period.

When the thermal conductivity and/or specific heat of some materials are defined as temperature dependent functions, those properties are adapted before each new iteration cycle (of the steady-state system for the first time step) and each next iteration step based on the obtained temperatures in the previous iteration cycle or previous time step.

Automatically calculated thermal properties are recalculated (if required by the corresponding setting in the **Calculation Parameters dialog box**, see D.2.4) before each iteration cycle or time step, and can be reported in the output report (see variables of type λ_{eq} and h_c in the report definition; cf. section B.24), but the initial values in the **Colours window** are not updated (as opposed to what happens in TRISCO, where the recalculated values are refreshed in the **Colours window**).

D.7 Unlinking air temperatures in cavities of type BC_FRE_S or BC_FREE

A cavity or layer (filled with air or any other fluid) can be modelled using several (adjacent) blocks having different colours. These colours can have different thermal properties (e.g. surface heat transfer coefficient), but must have the same type (i.e. all BC_FRE_S or all BC_FREE). Otherwise the calculation is immediately stopped, with the error message "BC (colour ...) touches other BC of different type".

The default implementation is to assume that all these adjacent colours making up the cavity or layer are linked.

- If the cavity is modelled with colours all of type BC_FRE_S, then this means that all colours get the same (unknown) environmental temperature, which is calculated solving the system. Possible convective heat powers for the individual colours in the same air cavity (column Pc in the **Colours window**) are cumulated and allocated to the central convection node (having the single environmental temperature).
- If the cavity is modelled with colours all of type BC_FREE (only possible with the RADCON module), then the radiative heat transfer is calculated taking together all these colours forming one zone in which all view factors are calculated. The convective heat transfer is calculated with reference to one central convection node (having an unknown air temperature).

This default behaviour can be changed. An air cavity can contain more than 1 convection node by using several colours in that cavity and unlinking those colours. For each unlinked colour and for the remaining block of not unlinked colours in the same air cavity a separate convection temperature will be calculated using separate convective heat balances (based on the surface temperatures and convective heat transfer coefficients of the adjacent material surfaces, and on the injected convective heat power). The radiative heat balance (in case of BC_FREE cavity) remains unchanged, i.e. considering the global air cavity in which all view factors are determined.

A colour can be manually unlinked by adding per colour (that you want to unlink) a text line at the end of the VOLTRA data file (with extension .vtr) using a text editor (e.g. Windows Notepad). The syntax of this line is:

unlk=c

where c = colour number (0..255) for unlinked colour.

When considering ventilation flows through different air zones (cf. section B.23 and D.8), the air temperatures of these zones are automatically made unlinked.

D.8 Processing ventilation flows

When ventilation flows are present in the input data, the system matrix is not symmetrical anymore, and therefore each iteration cycle is split in two successive subtasks:

- First the solution of a small subsystem of only convection nodes into which ventilation flows are injected. The number of these ventilation nodes is always less than or equal to the number of colours of type BC_FRE_S or BC_FREE (so ≤ 256). The surface temperatures that affect the convection nodes are considered as fixed boundary condition values (as obtained by the solution of the second subtask).
  The ventilation subsystem has an unsymmetrical system matrix, but has a limited number

of unknowns (i.e. the temperatures of the ventilation nodes, always ≤ 256), and is solved using a direct solution method.

- Secondly the solution of the ventilation replaced system, i.e. the whole system in which the calculated temperatures of the ventilation nodes are considered as fixed boundary conditions. This system has a symmetrical system matrix, and can be solved using an optimised iterative method.
  The calculated surface temperatures adjacent to ventilation zones are taken as fixed boundary condition values in the next ventilation subsystem to solve.

In the very first iteration cycle (i.e. of the first time step) no initial values of the surface temperatures are known (as required by the first subtask). The complete system is then solved without any ventilation flows to get start values for the surface temperatures for the first subtask of the next iteration cycle.

When ventilation flows are present, a variable number of iteration cycles is executed per iteration step independent of the given maximum number of iteration cycles as defined in the **Calculation Parameters dialog box**. The number of iteration cycles per iteration step is automatically stopped when all temperature differences between the calculated temperatures in the ventilation nodes of the current and previous iteration cycle are smaller than the given maximum temperature difference in the **Calculation Parameters dialog box** (a typical value is 0.0001°C, see D.2.2). However it may be possible (dependent on the input data) that there will be no convergence. In this case the iteration cycles are also stopped when any temperature difference of a ventilation node between the current and previous iteration cycle is greater than 1000000°C. The program resets all node temperatures of the whole system to 0°C, and the user is prompted to inspect and alter the input data.

D.9 Solar radiation processing

The solar processor is activated when:

1. A solar zone is present: i.e. a colour of type BC_SIMPL or BC_SKY with sun flag set (defined in **Colours window**, column "Sun", toggle value set to YES).
2. Solar radiation data are present: both a function G01 and a function D01 must be defined in the **Functions window** (B.12) in case of a real sun path (B.18.1); and a function I01 must be defined in the **Functions window** (B.12) in case of a fixed sun position with variable radiation (see B.18.2).

   G01 refers to a function for the global solar radiation (in W/m^2) on a horizontal surface.
   D01 refers to a function for the diffuse solar radiation (in W/m^2) on a horizontal surface.
   I01 refers to a function for the total solar radiation (in W/m^2) in a user-defined direction.

Input parameters with respect to the solar processor:

- The solar data as defined in the Solar Data dialog box (see B.15)
- Per material: solar reflection factor and solar transmission factor (see D.9.1).
  These material properties are defined in the last two columns of the **Colours window** (see B.10).
  A solar reflection factor function is referenced by R##, with ## = 01 to 99, and refers to the **Functions window** (see B.12.2).

- Calculation start day and time (*Edit → Calc Parameters...*, cf. section D.2.1). A calculation may be preceded by a pre-calculation period (to get dynamic start values of the node temperatures).
- Sun obstacles: shadow zones of other objects around the input bitmap, defined by spherical angular coordinates in the horizontal coordinate system (*Edit → Sun Obstacles*, cf. section B.19).

At every time step the solar load on the object model is calculated (see D.9.2 for external surfaces, D.9.5 for internal surfaces, and D.9.6 for opaque surfaces behind transparent materials). The solar load is expressed as a calculated node power (in W) per system node. These node powers are used as supplementary boundary conditions of the thermal system besides the other boundary conditions (e.g. the external temperature).

D.9.1 Definition of solar properties ρs and τs for MATERIAL

For each colour of type MATERIAL, the solar reflection factor (ρs) and the solar transmission factor (τs) need to be defined in the **Colours window** (see B.10).

For incident direct solar radiation the actual value of ρs may be dependent on the angle of incidence, if defined by a solar reflection factor function referenced by R## (see B.12.2).

There are two important calculation aspects to consider when quantifying these solar properties as inputs for a colour of type MATERIAL:

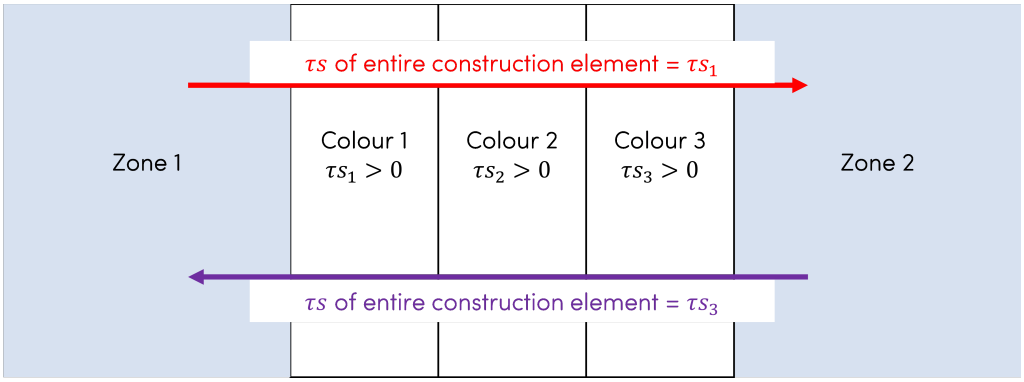
1. The solar reflection factor (ρs) and solar transmission factor (τs) are applied at the contact of a surface with a zone.

   A *zone* in this context is either a solar zone (BC_SIMPL or BC_SKY) or an internal zone with detailed radiation calculation (BC_FREE, BC_SKY, BC_NOSKY or TRANSMAT).

   If a colour of type MATERIAL in contact to the source zone from which the radiation enters has a solar transmission factor τs equal to 0, it is an *opaque material* (see D.9.5).

   A *transparent construction element* in VOLTRA consists of a single colour of type MATERIAL with $\tau s > 0$ or a system of adjacent transparent colour layers of type MATERIAL with $\tau s > 0$ or of type EQUIMAT (always assumed $\tau s = 1$).

   Within a system of adjacent transparent colour layers from a zone up to the next zone, only ρs and τs at the outer surface of the first colour layer (in contact to the source zone from which the radiation enters) are considered. τs can thus be understood as a property of the entire system of adjacent material layers and not just the material layer in contact to the source zone, it is good practice to give all adjacent enclosed material layers the same τs value in the **Colours window**.



A system of adjacent transparent material layers can alternatively be placed between a source zone from which the radiation enters and an opaque material (τs = 0). This is discussed in D.9.6.

2. The solar transmission factor τs in VOLTRA is applied *after* subtraction of the solar reflection (which can be dependent on the angle of incidence of solar radiation).

   Typically, the solar reflection, transmission and absorption are defined relative to the total incoming radiation, so

   $\rho + \tau + \alpha = 1$

   Or for normal incidence (angle of incidence 0°):

   $\rho_0 + \tau_0 + \alpha_0 = 1$

   However, for the input value of τs in VOLTRA, after subtraction of reflection ρs, applies:

   $\tau s + \alpha s = 1.$

   The input value of τs in VOLTRA can be calculated from the typical solar properties at normal incidence (angle of incidence 0°) ρ_0 and τ_0:

   $\tau_0 = (1 - \rho_0) \cdot \tau s$ or $\boxed{\tau s = \frac{\tau_0}{1-\rho_0}}$

D.9.2 Irradiance: solar load to external surfaces

An *external surface* is a surface of an object (with colour type MATERIAL) in direct contact with a *solar zone* (colour type BC_SIMPL or BC_SKY, and sun flag set in column "Sun" in the **Colours window**).

The position of the sun is calculated at the current simulation time (day of year, clock time) and the geographical location (latitude, longitude, time zone)[7]. Daylight savings time is not taken into account. The position of the sun is located by two angular measures: azimuth (horizontal clockwise angle between the sun position and south) and altitude (angle between the sun position and the horizon). The azimuth of the sun at a fixed clock time may differ for different days of the year due to the irregular movement of the sun (defined by the equation of time): e.g. the azimuth at 12 a.m. solar mean time for every day fluctuates around 0° (= south direction).

For the current time step the actual global solar radiation and diffuse solar radiation (in W/m^2) on a horizontal surface are derived from the corresponding external data files (see B.12.2) or weather data files (see B.12.3). Typically these weather data files contain hourly values.

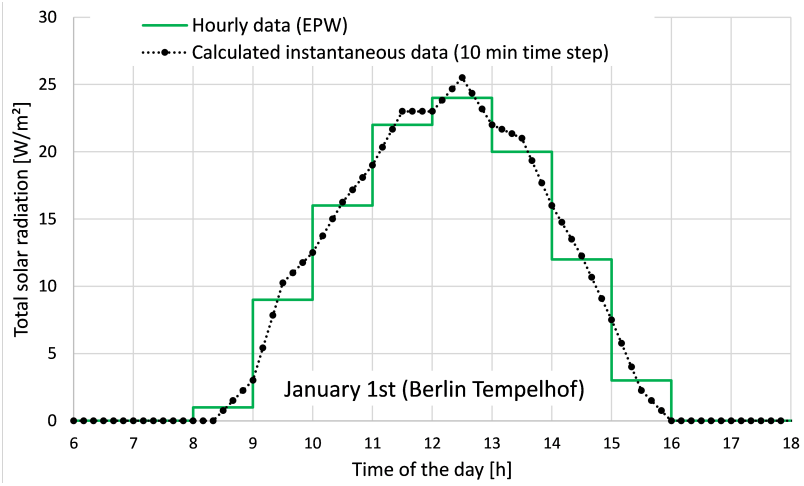
Values at simulation times are derived through simple linear interpolation in case of an external data file with linear function format (see B.12.2).

In case of an external data file with step function format (see B.12.2) or a weather data file (see B.12.3), the values in the external data file are converted to instantaneous values at simulation times using a mathematical algorithm[8]. This algorithm takes the empirically

[7] Roy G. et al., A note on solar declination and the equation of time, Architectural Science Review, Vol. 32, p. 43-51 (1989)

[8] McDowell T. et al., A new method for determining sub-hourly solar radiation from hourly data, Proceedings of the 2018 Building Performance Analysis Conference and Simbuild, p. 518-252 (2018)

calculated times of sunrise and sunset into account and produces instantaneous solar radiation values for any defined time step. The resulting solar radiation profiles ensure that the total solar radiation received matches the information in the data file, and have 'realistic' (though obviously estimated) shapes without discontinuities.



All produced instantaneous solar radiation values at simulation times are used in VOLTRA as average values over the simulation time step, where the simulation time is understood as the middle of the simulation time step (since the solar angles are calculated at each simulation time).

For each external surface the angular position against the north direction is computed: orientation (compass direction) and tilt (height above horizon). From the global and diffuse solar radiation on a horizontal surface, the direct and diffuse solar irradiance on the area around each node of the external surface is calculated.

The direct solar irradiance is derived from the direct solar radiation on a horizontal surface and the position of the sun. If the sun is not hidden by any sun obstacle (as defined in the **Sun Obstacles window**), a test ray from the surface node towards the sun is inspected on possible intersections with other (non-transparent) object surfaces (present in the bitmap). When there are no intersections, the node is sunlit. Otherwise the node is shaded and the direct solar irradiance on the area around the node is cancelled.

The estimation of the diffuse solar irradiance on the area around the node of the external surface is more complex.

- First the sky visibility from the elementary area around the node is used as reduction factor of the diffuse radiation. The sky visibility is calculated using a fixed number of test rays from the node with random direction on the hemisphere above horizon. This number of test rays is defined by the calculation parameter "Number of visibility rays between radiative surfaces" (see D.2.3). For all directions at the front side of the surface, the sky visibility is estimated as the fraction of the test rays not shaded by any sun obstacle or other (non-transparent) object in the bitmap. This is a good estimation of the view factor to the open sky from the elementary area around the node.
- Secondly the diffuse radiation depends on the tilt of the surface.
  In an isotropic diffuse model the diffuse radiation on a sloped surface is proportional to the square of the cosine of half the tilt angle. E.g. the diffuse radiation on a vertical surface is half the diffuse radiation on a horizontal surface (i.e. the surface only sees half of the sky

dome).
VOLTRA uses the more correct anisotropic diffuse model developed by Muneer[9], which also distinguishes between shaded and sunlit surfaces and between overcast and non-overcast conditions of the sunlit surfaces. The required parameters can be derived from sun position, slope, orientation, day of year, sunlit status, sky visibility, global and diffuse solar radiation data on a horizontal surface.

- Finally radiation reflected from the ground in front of the surface, must be added. This radiation is assumed isotropic, and is a function of the global solar radiation on a horizontal surface, the tilt of the surface, and the ground reflection factor, which is an input parameter in the Solar Data dialog box (normal value = 0.2). The ground reflected radiation on a vertical surface is half the ground reflected radiation on a horizontal downward surface.

If the solar zone is of type BC_SIMPL, then the reflected (direct or diffuse) solar radiation from an external surface is further ignored. If the solar zone is of type BC_SKY, this reflected solar radiation is further processed (see D.9.4).

D.9.3 Absorption of solar radiation

For an opaque external surface (with solar transmission factor equal to 0, see D.9.1), the radiation that is not reflected is absorbed. The absorbed solar radiation is converted to node powers in the surface nodes.

For a transparent external surface (with solar transmission factor greater than 0, see D.9.1), some part of the solar radiation is transmitted to internal zones (see D.9.5) or to an opaque surface behind the transparent construction (see D.9.6). The remainder of the solar radiation is absorbed.

The absorbed solar radiation is uniformly distributed (proportionally to the mesh distances) along the grid line through the surface node and perpendicular to the external surface, up to the opposite contact surface with an internal zone or opaque surface.

The report output qsol or absorbed solar flux (see B.24) is always given for the colour for which the relevant solar property τs is defined (see D.9.1). In case of a transparent construction element consisting of a number of adjacent transparent colour layers, only the colours in contact to the source zone from which the radiation enters thus will have non-zero qsol output, even though the absorbed radiation is distributed over the entire thickness of the transparent construction element consisting of a number of adjacent transparent colour layers for the thermal solution.

D.9.4 Processing of reflection

Reflection is only processed in zones with detailed radiation calculation (BC_FREE, BC_SKY, BC_NOSKY or TRANSMAT), so not in solar zones of type BC_SIMPL.

For direct radiation, if the reflection factor ρs of the surface is angle dependent (cf. definition of the material colour in the **Colours window**, see B.10), then this factor is determined through interpolation between the values in the solar reflection function file for the actual angle of incidence (angle between direct solar radiation and surface normal).

[9] Muneer T., Algorithms for estimating hourly solar irradiation on slopes, Building Serv. Eng. Res. Technol. 10(2), p. 81-83 (1989)

The reflection factor for diffuse radiation is the weighted mean value of the reflection factors for all possible angles of incidence.

Diffusely reflected radiation originates from either incoming diffuse radiation or incoming direct radiation. This reflected radiation is ideal diffuse (following Lambert's cosine law) and is again reflected, transmitted and absorbed. The final contributions of additional absorption on all surfaces are calculated using the radiosity method.

The radiosity of each elementary surface of the view factor grid equals the amount of energy that is given off. The energy incident on the surface (which we consider as unknown of the system) equals the sum of all reflected and transmitted energy from other surfaces, multiplied by the view factors from those surfaces to the first surface. The reflected energy from another surface equals the energy incident on that surface (which is an unknown of the system) multiplied by the (diffuse) reflection factor plus the known reflected energy (at the initial state of the radiosity system). The transmitted energy equals the energy incident on the opposite side of that wall (which is an unknown of the system) multiplied by the transmission factor at that side. The transmission factor must first be corrected for the reflection at that side (because the transmission factor in VOLTRA is defined after deduction of reflection, see D.9.1). Finally this leads to a set of linear equations.

The radiosity method implies that every time step a supplementary linear system of equations must be formulated and solved with the number of unknowns equal to the number of view factor surfaces of the view factor grid. This system is global for all zones (based on view factors), because these zones may be linked with one another via transparent walls.

D.9.5 Transmission to internal zone: solar load to internal surfaces

If the transmitted solar radiation reaches a boundary condition of type BC_FREE, BC_SKY, BC_NOSKY or TRANSMAT, then the corresponding zone is called an *internal zone*, through which the solar radiation is further propagated[10]. The material surfaces bordering an internal zone are *internal surfaces*. A zone of type BC_FREE has view factor based radiation and convection with a balanced unknown air temperature. This type is normally used for rooms or larger air cavities.

A zone of type TRANSMAT has view factor based radiation and conduction using a given thermal conductivity (of still air or another gas). This type is normally used for smaller cavities, e.g. the cavity of a double glazing.

In general it is recommended to use a view factor grid that is less detailed (i.e. with a smaller number of wider meshes) than the calculation grid. The view factors in the calculation grid are derived from the view factors in the view factor grid through scaling with the areas of the elementary surfaces. The precision is nearly the same, but the calculation speed can be enormously improved (see D.3).

The transmitted direct solar radiation in a node of an internal surface is projected along the direction of direct sun light (without refraction) upon the closest internal surface (of the same internal zone opposite to the emitter internal surface), forming the solar spot. The power of the transmitted direct solar radiation corresponding to the elementary grid surface around

[10] Note that if the colour behind the transparent construction is of type BC_SIMPL, the transmitted solar radiation cannot be accounted for because BC_SIMPL is not a view factor zone. The amount of solar radiation that would be transmitted is thus disregarded in the thermal calculation.

the emitter node is attributed to the elementary grid surface containing the intersection point of the projected solar beam. When this power would cause a flux (i.e. power divided by area) that is greater than the solar radiation on an external surface with the same orientation, then this power excess is spread over adjacent elementary grid surfaces with the same orientation. This smoothing occurs when the area of the receiver surface is (much) smaller than the area of the emitter surface. Therefore, it is recommended to use a (more or less) equidistant grid.

The direct solar radiation through transparent materials (e.g. window glass) causes a sunlit spot on the internal surfaces of an internal zone. This spot will be sharper when the calculation grid is finer.

Part of the incident direct solar radiation on an internal surface is reflected, using the reflection factor of the surface material colour (see D.9.1). This reflection factor may be dependent on the angle of incidence (when the reflection factor is defined by a reflection factor function). The reflected radiation is treated as described in section D.9.4.

The remainder of the incident direct solar radiation on the internal surface is transmitted and absorbed. The transmitted and absorbed direct radiation are treated in the same way as transmitted and absorbed direct solar radiation on an external surface (cf. section D.9.2 and D.9.3).

The transmitted diffuse solar radiation of an elementary grid surface is projected upon the other elementary grid surfaces (of the same internal zone) proportional to the view factors from the emitter surface to the receiver surfaces.

Part of the incident diffuse solar radiation on an internal surface is reflected, using the reflection factor for diffuse radiation. The reflected diffuse radiation is treated as described in section D.9.4.

The remainder of the incident diffuse solar radiation on an internal surface is transmitted and absorbed analogously to transmitted and absorbed diffuse solar radiation on an external surface (cf. section D.9.2 and D.9.3).

D.9.6 Solar load to opaque surfaces behind transparent materials

In previous versions of VOLTRA (VOLTRA v8.1 and before), in case a transparent construction element was placed in front of an opaque colour (τs = 0), the radiation transmitted through the transparent construction element was ignored. If the user wanted to account for absorption and reflection of radiation by the opaque surface behind a transparent construction element, a very thin layer representing a virtual zone needed to be drawn manually between the transparent construction element and the opaque surface. From VOLTRA v9, this manual work-around is no longer necessary, as it is integrated in the calculation.

An infinitely thin gap is assumed in the calculation between a transparent construction element and an opaque material surface, with view factor between both opposite surfaces (which is the contact surface) equal to 1.

These infinitely thin gaps are treated as additional view factor zones, thus similar to internal zones, as described in section D.9.5.

The direct and diffuse transmitted solar radiation that hits an opaque material behind a transparent material is added to the radiosity system for redistribution of diffuse radiation,

based on diffuse reflection, absorption and transmission coefficients of the two opposite surfaces. The direct transmitted solar radiation is thus considered as additional diffuse transmitted solar radiation, so that it can be included in the radiosity system.

The absorbed radiation at the opaque material surface, resulting from the solution of the radiosity system, is then converted to system node powers on the opaque material surface.

D.10 Heat flow divergences

The quality of the solution is mainly determined by the heat flow divergence of the total object, called in short object divergence. It is defined as the sum of all heat flows (positive and negative) entering the object, divided by half the sum of the absolute values of all these heat flows:

$$OD = \frac{|\Phi_{in,o} - \Phi_{out,o}|}{0.5 \cdot (|\Phi_{in,o}| + |\Phi_{out,o}|)} \cdot 100$$

with	OD = object divergence	[%]
	$\Phi_{in,o}$ = total incoming heat flow for the object	[W]
	$\Phi_{out,o}$ = total outgoing heat flow for the object	[W]

By using the factor 0.5 in the denominator, the denominator can be interpreted as the heat flow "passing" the object.
The maximum possible object divergence is 200 %, i.e. when there are only incoming heat flows and no outgoing heat flows: $\Phi_{in,o} > 0$ and $\Phi_{out,o} = 0$ (or vice versa $\Phi_{in,o} = 0$ and $\Phi_{out,o} > 0$).

An object divergence tolerance (cf. calculation parameter “Maximum heat flow divergence for total object”, see D.2.2) of 0.001 % is recommended as reliable stopping criterion. According to EN ISO 10211 the object divergence should be less than 0.1 %.

In VOLTRA also a second heat flow divergence is used as stopping criterion: the maximum heat flow divergence for all nodes, called in short node divergence. It is defined in a similar way as the object divergence:

$$ND = \max_n \left(\frac{|\Phi_{in,n} - \Phi_{out,n}|}{0.5 \cdot (|\Phi_{in,n}| + |\Phi_{out,n}|)} \cdot 100 \right)$$

with	ND = node divergence	[%]
	$\Phi_{in,n}$ = total incoming heat flow for the control volume around node n	[W]
	$\Phi_{out,n}$ = total outgoing heat flow for the control volume around node n	[W]

In VOLTRA the default node divergence tolerance is 1 % (cf. calculation parameter “Maximum heat flow divergence for any node”).

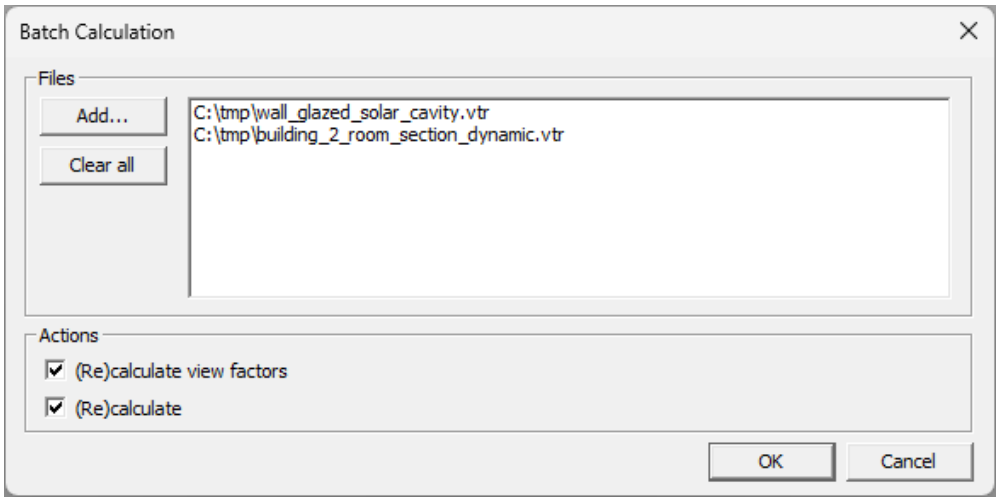
Sometimes it happens that the node divergence does not decrease during the calculation. This is caused by an “unlucky” node, that is connected to its neighbouring nodes in an “unfavourable” way (due to obtuse triangles and/or strongly different thermal conductivities). Then a higher node divergence tolerance should be set.

Cancel the node divergence test by using a node divergence tolerance of 201 % (which is higher than the theoretical 200 %, to be safe for possible numerical rounding errors).

D.11 Batch calculation

Menu command: *File → Batch Calculation...*

In the Batch Calculation dialog box you can choose which .vtr files to handle, and which actions to execute.

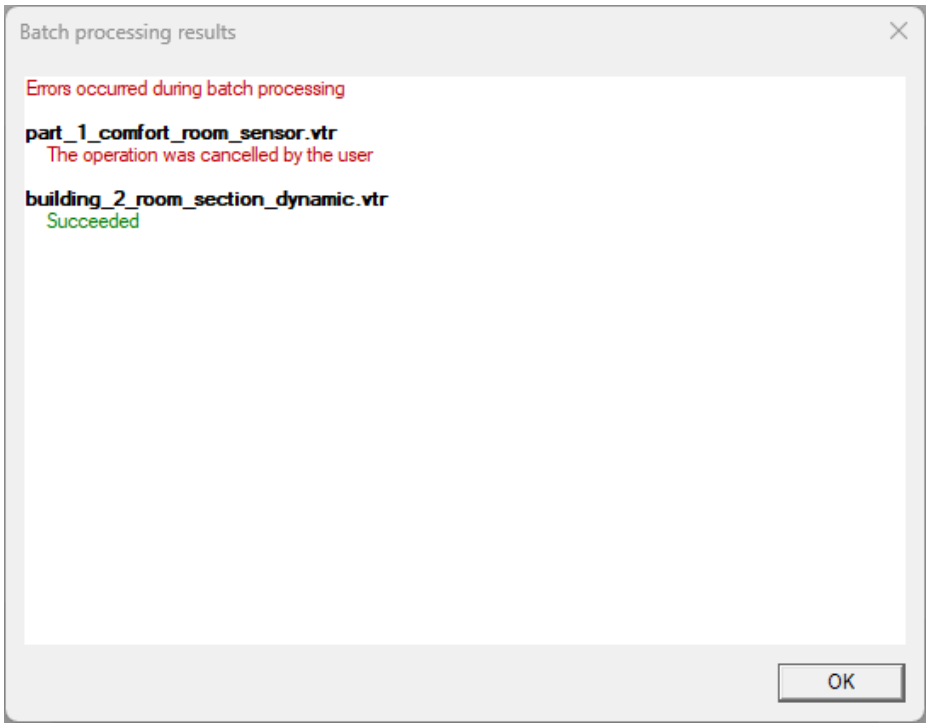


Add .vtr files by pressing the Add button and selecting one or more .vtr files from the same data file folder (hold down the <Ctrl> key or <Shift> key to select more than one file). You can do this multiple times to select files in different directories.

Select one or more actions to execute on each file:

- *(Re)calculate view factors*: forces the calculation of the view factors, even if they are already available in a .vwf file (only for models with radiation zones).
- *(Re)calculate*: If checked the thermal calculation is done, as with the command *Calc → Calc System*. Without *(Re)calculate view factors*, the view factors are only calculated if they are needed and no .vwf file is available.

All files are processed sequentially. At the end of the run, a summary is shown.



Per data file a report file is written to disk, containing the output report corresponding to the report definition of the data file (cf. sections B.24 and B.25). The report file has the same file name as the data file, and has the extension .txt.

When graphic output (i.e. snapshots at given time points and/or animation files of given time periods) is wanted afterwards, the data file should specify that step solutions must be saved to disk (cf. section B.26).

D.12 Command line program execution

VOLTRA can be run from the command line prompt with a data file path as parameter to open that file.

Example of command line launching of VOLTRA and opening a file:

Run → "c:\...\Physibel\VOLTRA9\VOLTRA.exe" "..\Documents\Physibel\VOLTRA9\Demofiles\1 - SUMMER - SOLAR\flat_roof_solar.vtr"

Flags can be added to the command line prompt determining which actions need to be taken after opening the file, analogous to the choices offered in the Batch Calculation dialog box (D.11):

- "/viewfactors" to force the calculation of the view factors, even if they are already available in a .vwf file (only for models with radiation zones).
- "/calculate" for the thermal calculation. Without "/viewfactors", the view factors are only calculated if they are needed and no .vwf file is available.

Example of command line launch:

Run → "c:\...\Physibel\VOLTRA9\VOLTRA.exe" "..\Documents\Physibel\VOLTRA9\Demofiles\1 - SUMMER - SOLAR\flat_roof_solar.vtr" /viewfactors /calculate

Alternatively to adding flags, the legacy system adding the switch "/Automation" is still available. It is equivalent to "/calculate".

Then no further user action is required. The program starts and ends automatically, and the output is a report file and/or a series of graphic snapshots (as in a batch calculation, cf. section D.11).

Example of command line launch:

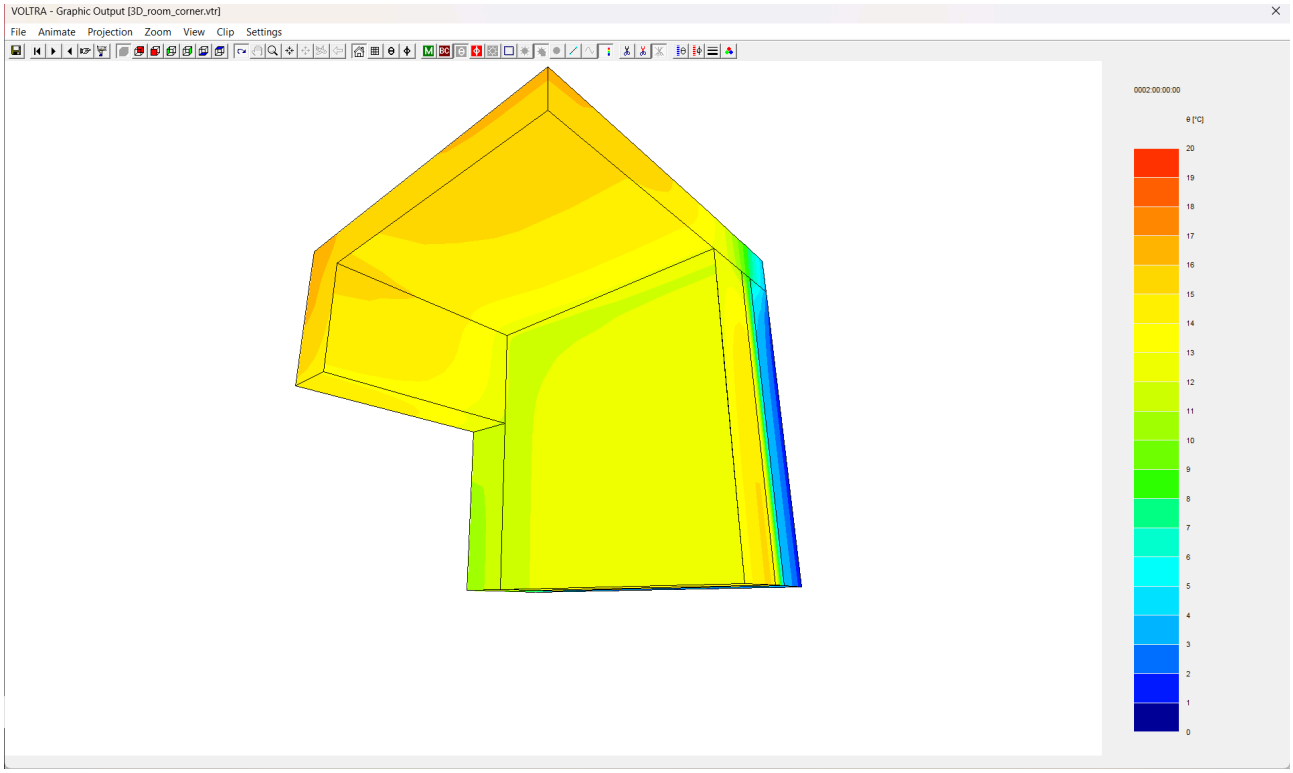
Run → "c:\...\Physibel\VOLTRA9\VOLTRA.exe" "..\Documents\Physibel\VOLTRA9\Demofiles\1 - SUMMER - SOLAR\flat_roof_solar.vtr" /Automation

CHAPTER E GRAPHIC OUTPUT

E.1 Graphic Output window

The **Graphic Output window** (which covers the complete VOLTRA application window) is opened with the command *Output → Graphic Output*.

To return to the main VOLTRA window, choose the command *File → Exit*.

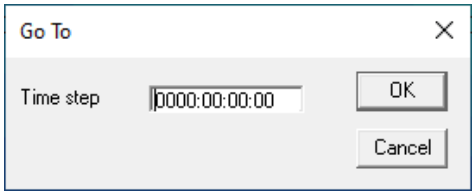


The material colour, boundary condition colour, temperature or heat flux (dependent on the current view option) under the mouse cursor point is shown in the status bar at the bottom of the **Graphic Output window**.

E.2 Animate functions

The animate functions are available through the Animate submenu.





The next functions are only available when solution files (*.sol) of a calculation are present on disk (see section B.26):

First Time Step	Loads the first solution file (for the start of the calculation duration, after the precalculation duration).
Next Time Step	Loads the next solution file.
Previous Time Step	Loads the previous solution file.

Go To...	Loads the solution file of a time step (of the calculation duration) entered by the user.
Play	Shows successively all solution files starting from the current time step. The playing can be stopped by pressing the <Esc> key.

E.3 Save animation images

Menu command: *File → Save Animation Images...*

A sequence of all animation images (as a result of the command *Animate → Play*) can be saved on disk as separate .png files.

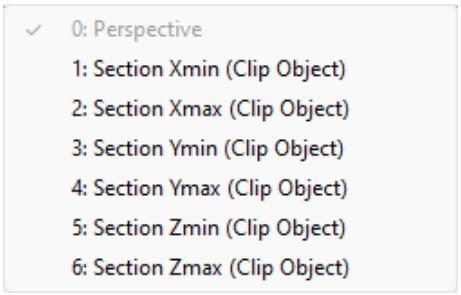
A Save As dialog box is opened to prompt for a file name.

The image size of saved bitmap and font size of legend text are defined via *Settings → Image Size...*.

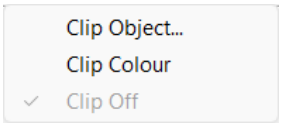
The solution files are loaded consecutively (started with the first solution at time 0000:00:00:00), shown in the **Graphics Output window** and saved. A 4 digit number (starting from 0000) is appended to the chosen file name to generate the different .png files. Saving animation images can be interrupted by pressing the <Esc> key.

E.4 Projections and clip object

Projection submenu

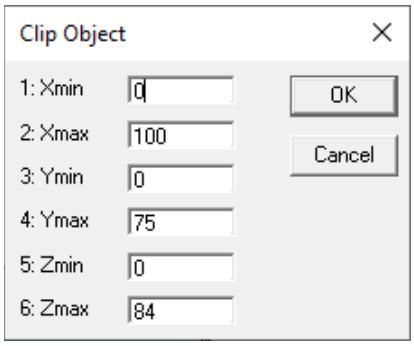


Clip submenu



- *Projection → Perspective*: same 3D perspective view as in **Image window** (see C.4).
- *Projection → Section ...*: section view through object along grid coordinate.

The clip object boundaries as defined in either the **Clip Object window** from the main window (cf. section C.10) or the Clip Object dialog box (*Clip → Clip Object...* from the **Graphics Output window**) are used as section definitions.
The view directions for the sections through the minimum and maximum clip coordinates are facing each other (i.e. viewing towards the clipped object).



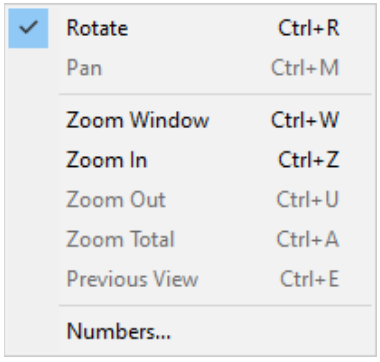
By editing the clip coordinates, cross sections through all grid planes are possible.
If a clipping plane does not intersect the object, then the corresponding section view is empty.

Individual colours can also be clipped from the object (*Clip → Clip Colour*), similarly as in the main window (C.11).

Reset all clip coordinates using *Clip → Clip Off*.

E.5 Zoom commands

Zoom submenu

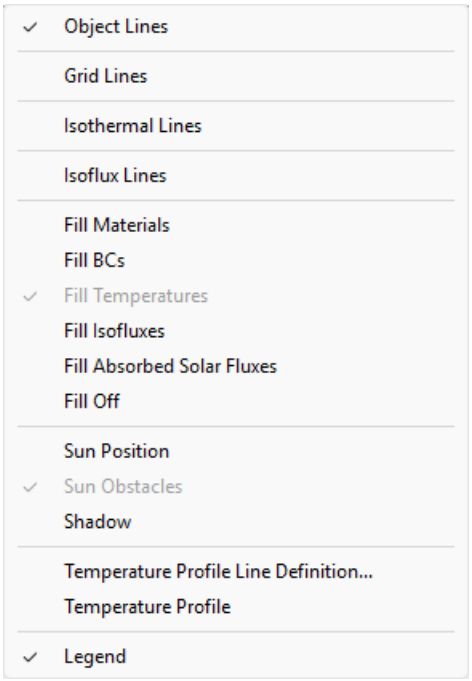


The zoom commands are identical to those of the **Image window** in the main menu (see C.3).

However a mouse drag movement in rotate or pan mode works not dynamically as in the **Image window** (because the **Graphic Output window** does not make use of OpenGL hardware acceleration). A rubber-band vector is drawn, and after releasing the left mouse button the new view is painted.

Zooming in and out is also possible via scrolling forward and backward of the mouse wheel.

E.6 View submenu



View → Legend: show a legend bar at the right side of the figure

Remark

For temperature dependent thermal conductivities the values that are shown in the legend (when the view option *View → Fill Materials* is chosen), are all taken at 10°C.

E.6.1 Lines

The first 4 options can be switched on or off independently:

- *View → Object Lines*: show object contour lines.
- *View → Grid Lines*: show grid lines on object surfaces.
- *View → Isothermal Lines*: show isothermal lines conform to the temperature range parameters (cf. section E.7).
  Only possible when a calculation solution is available.
- *View → Isoflux Lines*: show isoflux lines conform to the flux range parameters (see E.7).
  Only possible when a calculation solution is available.

E.6.2 Fill modes

One of the following fill modes is to be selected:

- *View → Fill Materials*: fill material surfaces using the object block colours (as defined in the **Colours window**).
  The status bar shows the colour number and name of the material surface under the mouse cursor point.
- *View → Fill BCs*: fill material surfaces using the adjacent surface boundary condition block colours (as defined in the **Colours window**).
  If no surface boundary condition is active on a material surface (i.e. an implicit adiabatic boundary condition is assumed), then the fill colour is light yellow.

The status bar shows the colour number and name of the boundary condition imposed on the material surface under the mouse cursor point.

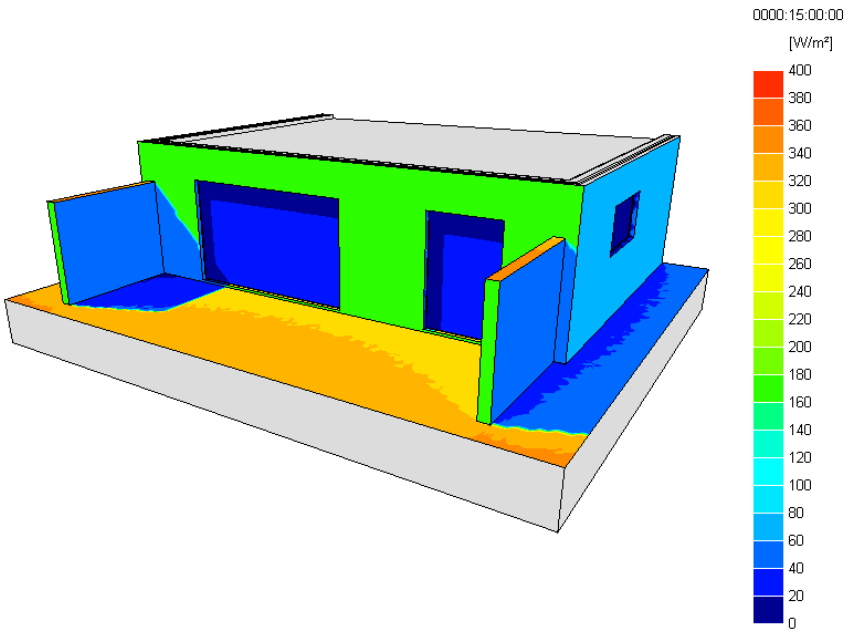
- *View → Fill Temperatures*: fill material surfaces using temperature colours conform to the temperature range parameters (see E.7) and of the selected thermal palette (see A.9 and E.7).

  The status bar shows the temperature of the material surface under the mouse cursor point.
  Only possible when a calculation solution is available.

- *View → Fill Isofluxes*: fill material surfaces using isoflux colours conform to the flux range parameters (see E.7).
  Only possible when a calculation solution is available.

- *View → Fill Absorbed Solar Fluxes*: when the solar fluxes are calculated (i.e. after *Calc → Calc Solar Fluxes* or *Calc → Calc System*) and the calculation step solutions (.sfd and .sfx files) are saved on disk (B.26), this option allows to visualise the absorbed solar fluxes on the object surfaces, conform the flux range (see E.7).

  For transparent surfaces the total absorbed solar flux is shown, before distributing the absorbed solar power into node powers within the wall behind the transparent surface (D.9.3).



- *View → Fill Off*: switch off filling.

E.6.3 Sun position and obstacles

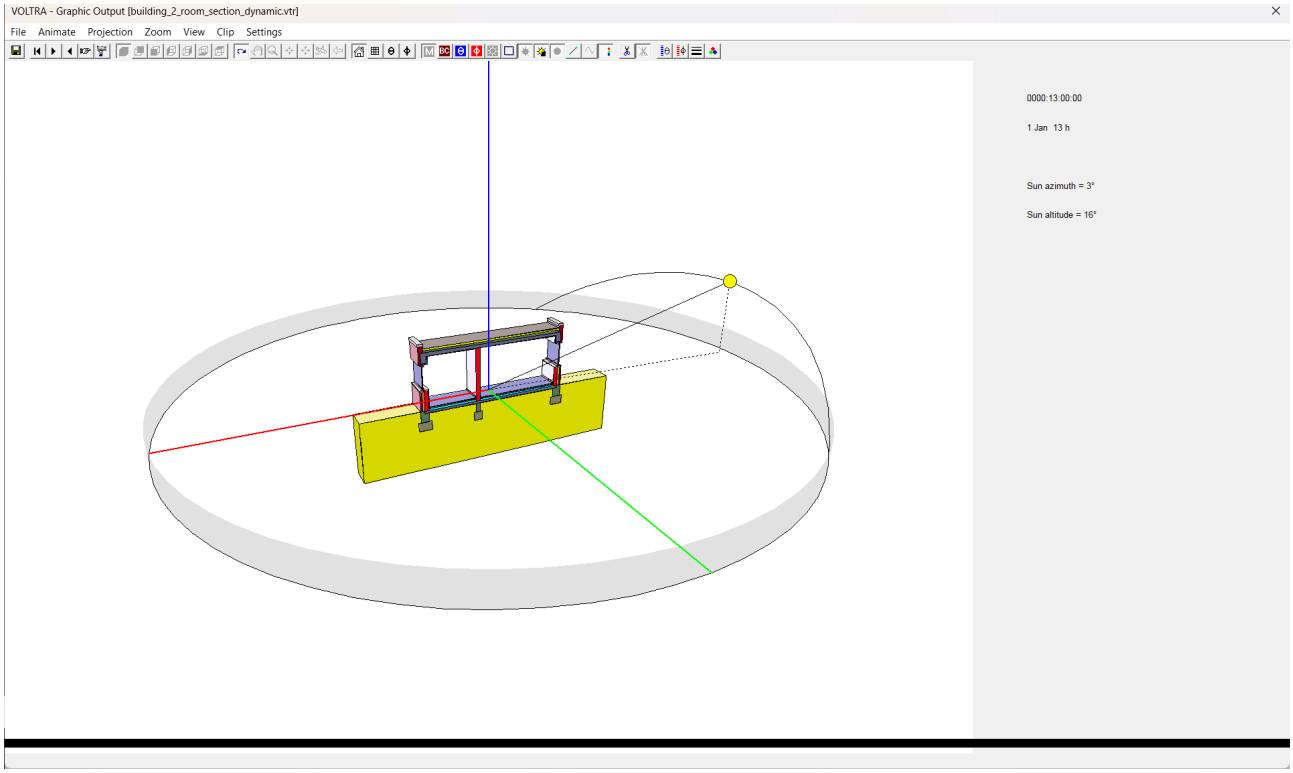
View → Sun Position

The sun position (azimuth and altitude) in the horizontal coordinate system (in which the plane XY coincides with the horizon, the X axis points to the north, the Y axis points to the west, and the Z axis points to the zenith) at any time can be viewed with *View → Sun Position*.

The parameters that define the position of the sun are:

- North orientation (*Edit → Solar Data...*).
- Geographic position (*Edit → Solar Data...*).
- Day of year at start of calculation (*Edit → Calculation Parameters...*).
- Time setting (*Animate → Go To...*).
  The start of calculation corresponds with time 0000:00:00:00.

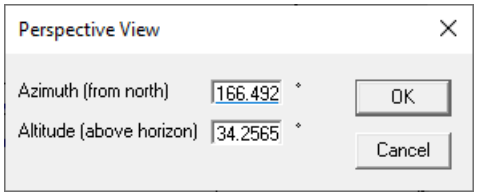
If the toggle *View → Sun Obstacles* is on, the sun obstacles as defined in the **Sun Obstacles window** (B.19) are shown in grey.



The 3D image can be rotated with *Zoom → Rotate* (Rotate mode is activated automatically by holding down the mouse wheel).

The perspective viewpoint (azimuth and altitude from the midpoint of the bitmap to the observer in the horizontal coordinate system) is entered with *Zoom → Perspective View...*.

All other zoom functions are suppressed while the option *View → Sun Position* is used.



Use the function *Animate → Play* to show the sun position changing with time.

The time step defined in the **Calculation Parameters dialog box** (*Edit → Calc Parameters...*) is used as animation time step.

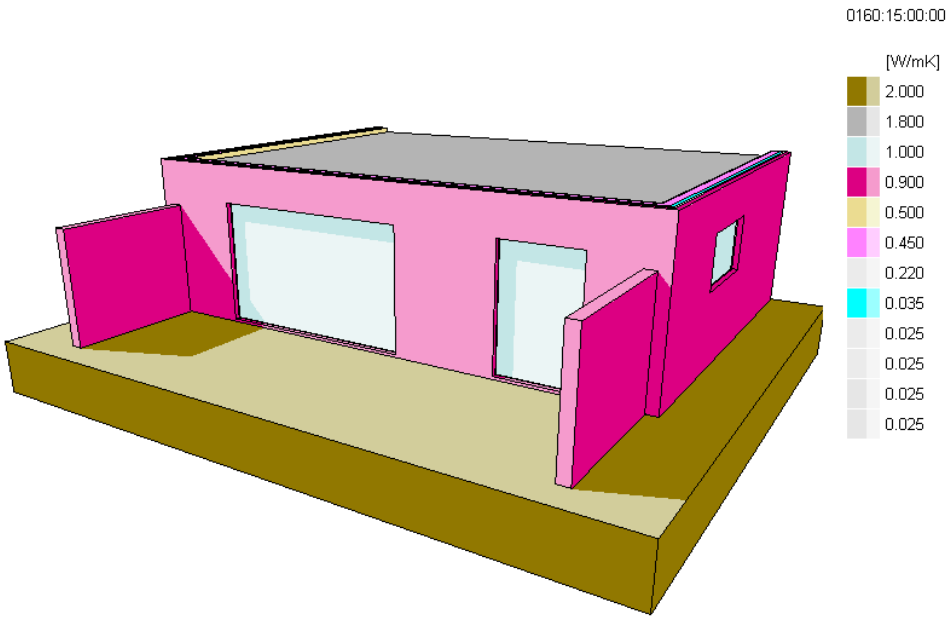
E.6.4 Sun shaded image

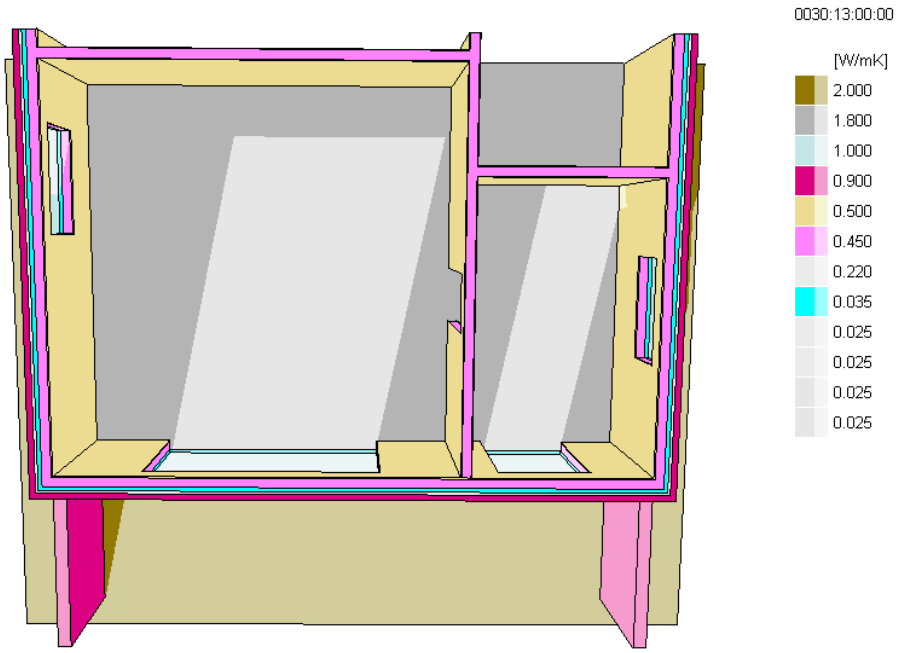
View → Shadow

When the sun is present in the object model (which requires the presence of a solar zone, i.e. a colour of type BC_SIMPL or BC_SKY with sun input parameter set, see B.10.8), a sun shaded image can be generated.

A sun shaded image can be generated before the system calculation is done. Sun obstacles (as defined in the **Sun Obstacles window**, B.19) are ignored.

Sunlit surfaces are shown on external surfaces bordering a solar zone, and on material surfaces of internal zones through transparent walls. If wanted, the object may be clipped, using *Edit* → *Clip Object* (as in the second figure below, which is a top view with the ceiling clipped off).



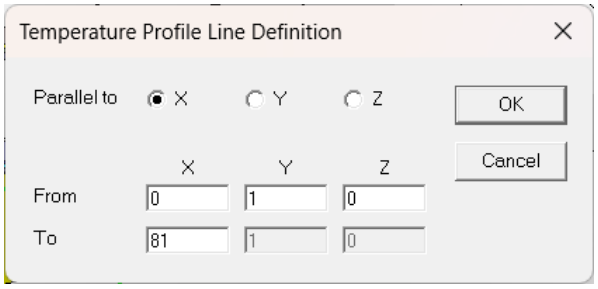


An animation with sun shaded images can be made using *Animate* ⟶ *Play*. The time step defined in the **Calculation Parameters dialog box** (D.2.1) is used as animation time step.

E.6.5 Temperature profile along a grid line

For any user defined grid line a temperature graph through the intersected material layers can be drawn per calculation time step.

First the desired grid line must be defined using the command *View* ⟶ *Temperature Profile Line Definition...*

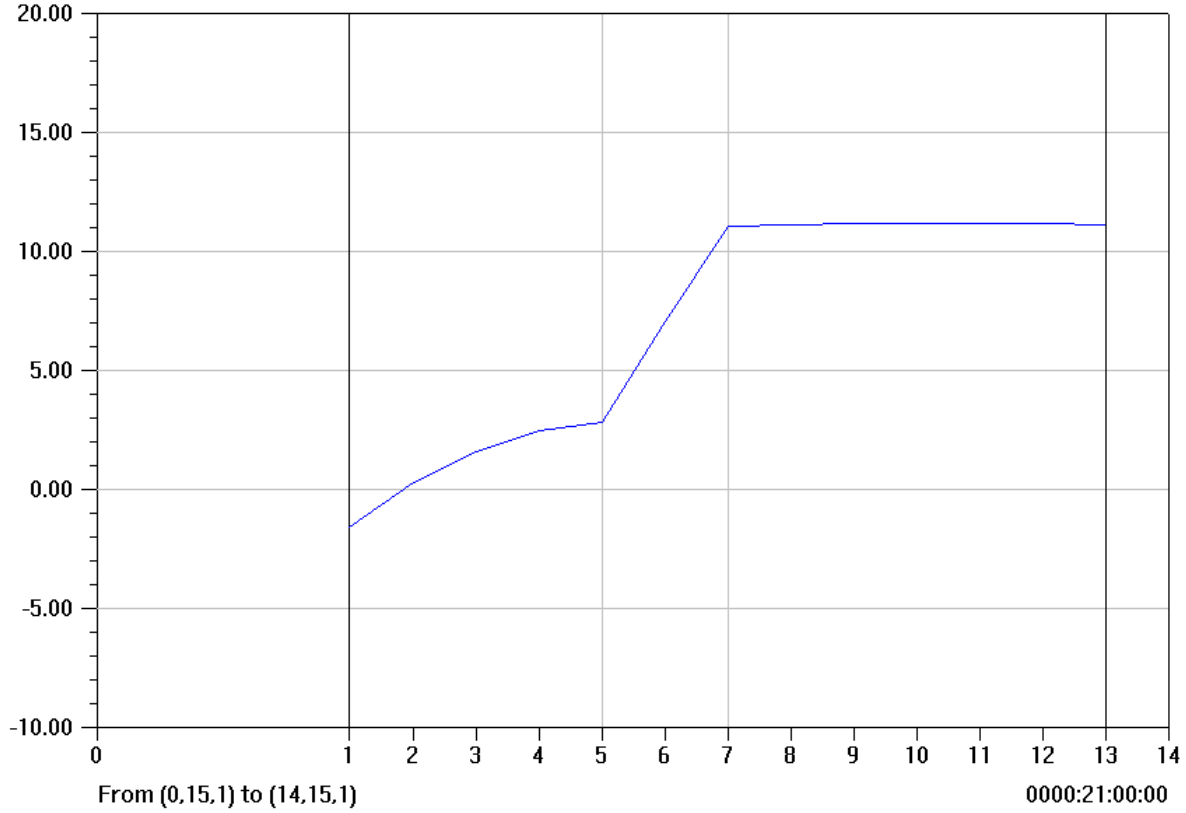


In the dialog box choose the grid line direction:

- X, Y or Z (in the XYZ coordinate system), or
- R, C or L (in the RCL coordinate system).

Then define the grid coordinates of the starting node (“from”) and ending node (“to”) of the line.

The toggle command *View* ⟶ *Temperature Profile* shows the temperature graph.



The horizontal axis shows the grid coordinates along the chosen grid direction.

The current calculation time step is displayed at the bottom right (and can be modified, e.g. using *Animate* ⟶ *Go To* ...).

The vertical axis shows the temperature scale conform to the temperature range parameters (see E.7).

The blue function line corresponds to the temperature solution at the current time step.

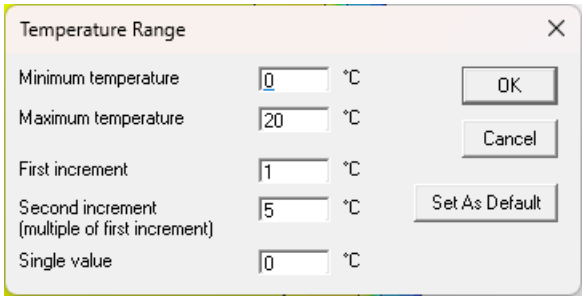
E.7 Settings



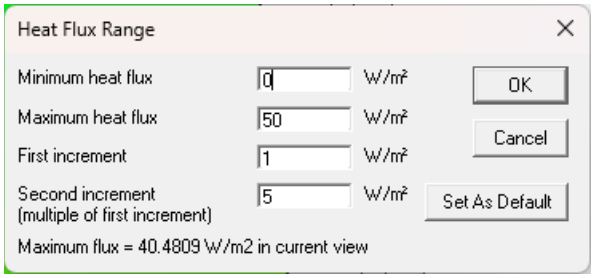
Settings ⟶ Temperature Range...

- Minimum temperature [°C] in graphic output.
- Maximum temperature [°C] in graphic output.
- First (or minor) increment [°C].
  Used for temperature filling (*View ⟶ Fill Temperatures*) and isothermal lines (*View ⟶ Isothermal Lines*).

- Second (or major) increment [°C] (must be a multiple of first increment).
  Only used with isothermal lines (*View → Isothermal Lines*).
- Single temperature value [°C] (e.g. dew point).
  Only used with isothermal lines (*View → Isothermal Lines*).



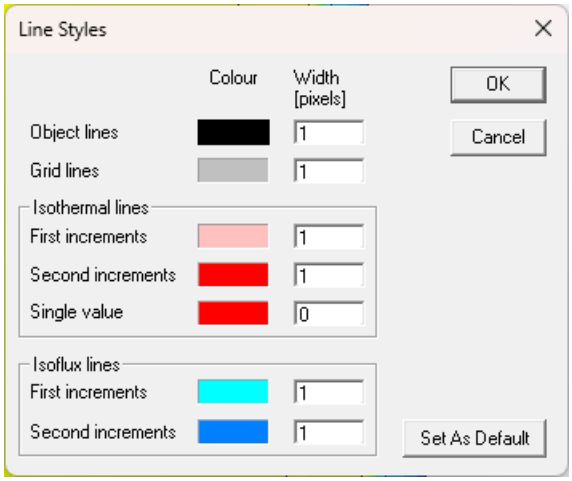
Settings → Flux Range...



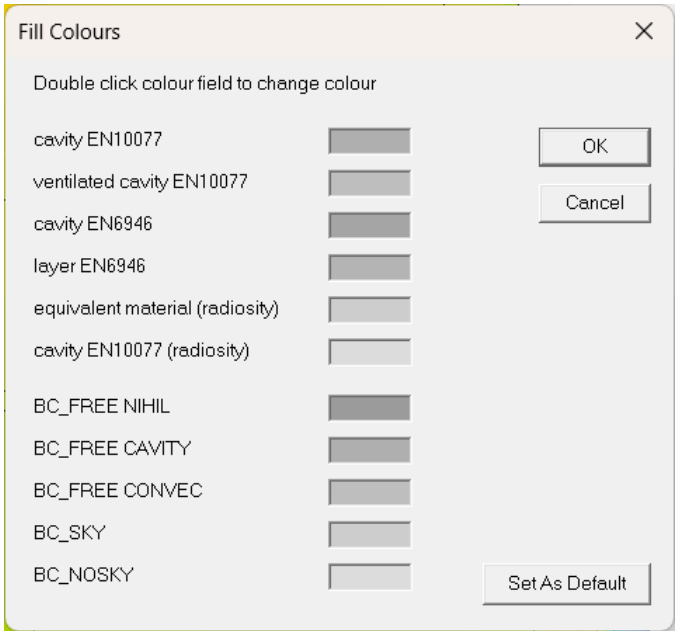
Settings → Line Styles...

A colour field can be edited by double clicking with the left mouse button, which opens a Colour dialog box in which the colour value (RGB components) can be defined.

The width of isothermal lines can be set to 0 pixels to hide them. This is useful when e.g. only the second isothermal lines are desired in the figure.



Settings → Fill Colours...



Colours with automatically calculated properties (see D.1) are clustered thematically in the **Graphic Output window** according to the categories of the Fill Colours dialog box, to avoid a potentially long list of colours of the same type (and subtype and standard) with small variations in the calculated thermal property. The fill colour of the clusters can be edited by double clicking on it with the left mouse button, which opens a Colour dialog box in which the colour can be defined.

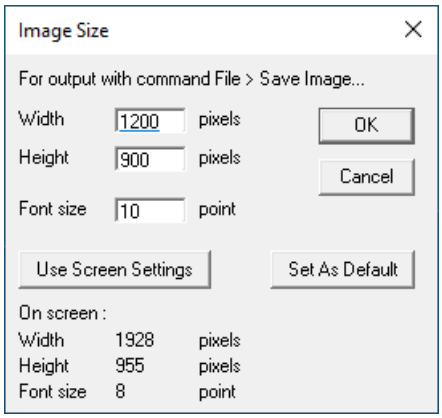
Settings → Thermal palette...



The selection of the thermal palette is synchronised with that selected in the main window (see A.9).

Settings → Image Size...

Set bitmap size for saving .png files (*File → Save Image...* and *File → Save Animation Images...*).



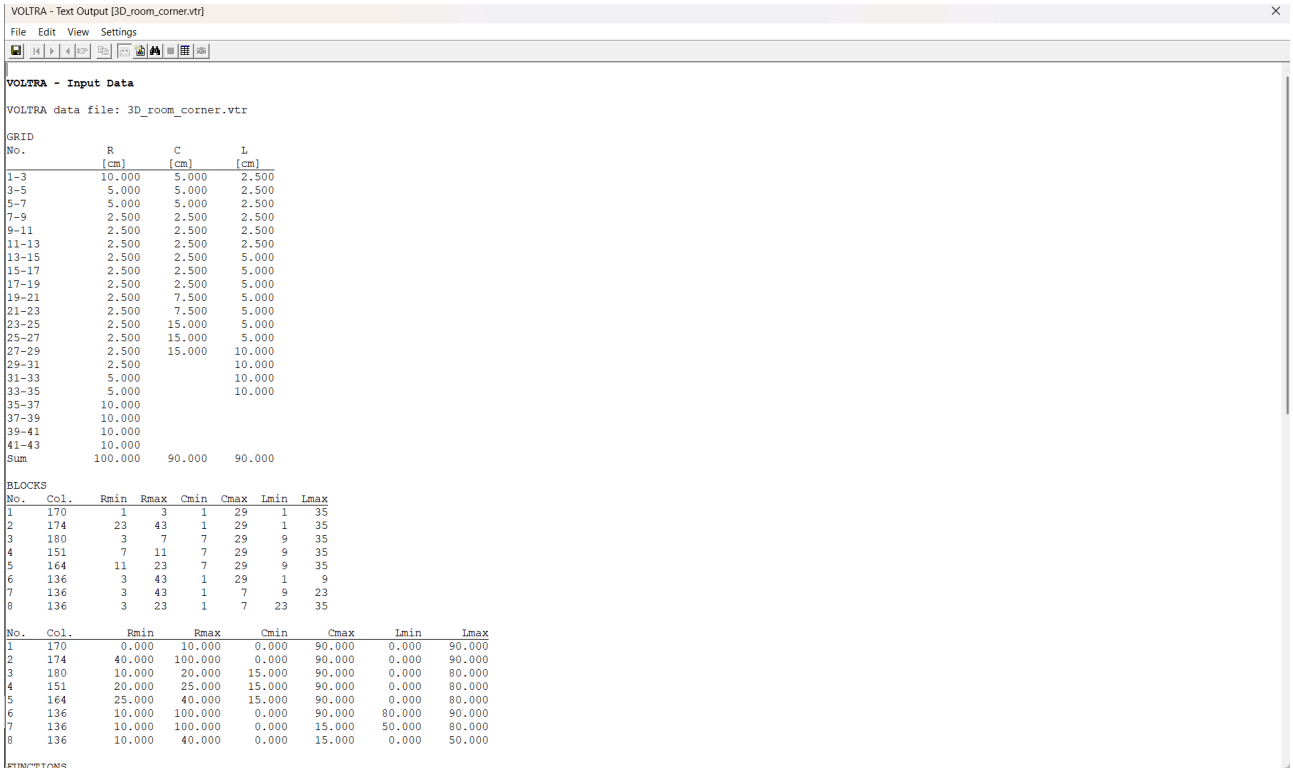
The font size refers to text characters in the legend or labels.

CHAPTER F TEXT OUTPUT

F.1 Text Output window

The **Text Output window** (which covers the complete VOLTRA application window) is opened with the command *Output → Text Output*.

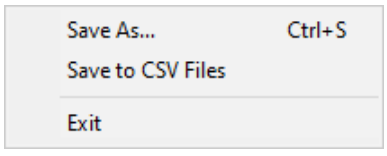
To return to the main VOLTRA window, press the <Esc> key or choose the command *File → Exit*.



The text within the **Text Output window** can be edited.

F.2 File commands

File submenu



- *File → Save As...*
  Save the whole text in the current **Text Output window** into a Microsoft Word document (extension .docx) on disk.
- *File → Save to CSV Files*
  Save the text output to .csv files for use in a spreadsheet program (e.g. Microsoft Excel).
- *File → Exit*
  Return to main VOLTRA window.

F.3 View options

F.3.1 Input data

The input data is shown in the **Text Output window** using the command *View → Input Data*.

VOLTRA - Input Data

VOLTRA data file: 3D_room_corner.vtr

GRID

No.	R [cm]	C [cm]	L [cm]
1-3	10.000	5.000	2.500
3-5	5.000	5.000	2.500
5-7	5.000	5.000	2.500
7-9	2.500	2.500	2.500
9-11	2.500	2.500	2.500
11-13	2.500	2.500	2.500
13-15	2.500	2.500	5.000
15-17	2.500	2.500	5.000
17-19	2.500	2.500	5.000
19-21	2.500	7.500	5.000
21-23	2.500	7.500	5.000
23-25	2.500	15.000	5.000
25-27	2.500	15.000	5.000
27-29	2.500	15.000	10.000
29-31	2.500		10.000
31-33	5.000		10.000
33-35	5.000		10.000
35-37	10.000		
37-39	10.000		
39-41	10.000		
41-43	10.000		
Sum	100.000	90.000	90.000

BLOCKS

No.	Col.	Rmin	Rmax	Cmin	Cmax	Lmin	Lmax
1	170	1	3	1	29	1	35
2	174	23	43	1	29	1	35
3	180	3	7	7	29	9	35
4	151	7	11	7	29	9	35
5	164	11	23	7	29	9	35
6	136	3	43	1	29	1	9
7	136	3	43	1	7	9	23
8	136	3	23	1	7	23	35

No.	Col.	Rmin	Rmax	Cmin	Cmax	Lmin	Lmax
1	170	0.000	10.000	0.000	90.000	0.000	90.000
2	174	40.000	100.000	0.000	90.000	0.000	90.000
3	180	10.000	20.000	15.000	90.000	0.000	80.000
4	151	20.000	25.000	15.000	90.000	0.000	80.000
5	164	25.000	40.000	15.000	90.000	0.000	80.000
6	136	10.000	100.000	0.000	90.000	80.000	90.000
7	136	10.000	100.000	0.000	15.000	50.000	80.000
8	136	10.000	40.000	0.000	15.000	0.000	50.000

```
FUNCTIONS
T01: SINE
  Mean value = 5°C
  Amplitudo = 5°C
  Period = 1440 min
  Offset = 360 min
T02: STEP2
  Level 1 = 20°C
  Level 2 = 10°C
  Period = 1440 min
  Duration level 1 = 720 min
  Transition = 120 min
  Start level 1 = 480 min
```

COLOURS

Col.	Type	Subtype	Phys. flow	Geom. flow	Name	eps1 / eps2 [- / -]	lambda [W/mK]	eps [-]
136	MATERIAL				reinforced_concrete_(stee		2.500	
151	MATERIAL				insulation_0.035_W/mK		0.035	
164	MATERIAL				masonry_semi-heavy		0.600	
170	BC_SIMPL	HE			exterior			
174	BC_SIMPL	NIHIL			interior_(normal)			
180	MATERIAL				masonry_heavy		0.900	

Col.	rho [kg/m3]	c [J/kgK]	t [°C]	h [W/m²K]	q [W/m²]	ta [°C]	hc [W/m²K]	Pc [W/m]	tr [°C]	Sun	rs [-]	ts [-]	Standard
136	2500.0	1000.0									0.10	0.00	
151	30.0	1000.0									0.10	0.00	
164	1350.0	840.0									0.10	0.00	
170			T01	25.00	0					NO			EN6946
174			T02	7.70	0					NO			NIHIL
180	1850.0	840.0									0.10	0.00	

OUTPUT NODES

No.	R	C	L
1	23	1	35
2	23	1	23
3	23	7	23
4	23	7	9
5	23	29	9
6	23	29	35

```
Calculation parameters
Time step interval = 0000:00:10:00
Start-up calculation duration = 0000:00:00:00
Calculation duration = 0002:00:00:00
Start of calculation = 0000:00:00:00
Iteration cycles = 5
Maximum number of iterations (within each iteration cycle) = 10000
Maximum temperature difference (within each iteration cycle) = 0.0001°C
Heat flow divergence for total object = 0.001 %
Heat flow divergence for worst node = 1 %
Automatic recalculation of CEN-values
Default temperature difference across airspace = 10°C
```

F.3.2 Results per time step

Main calculation results for a specific time step are shown using the command *View → Time Step Results*. This is only available for the time steps for which the solution is saved to disk (see B.26). The time step can be set using one of the first four commands in the Edit submenu, which are analogous as in the Animate submenu in the **Graphic Output window** (cf. section E.2).



First Time Step	Loads the first solution file (for the start of the calculation duration, after the precalculation duration).
Next Time Step	Loads the next solution file.
Previous Time Step	Loads the previous solution file.
Go To...	Loads the solution file of a time step (of the calculation duration) entered by the user.

VOLTRA - Calculation Results

VOLTRA data file: 3D_room_corner.vtr
Number of nodes = 4000

Time step 0002:00:00:00

Heat flow divergence for total object = 1.55984e-11 %
Heat flow divergence for worst node = 5.41413e-09 %

Col.	Type	Name	tmin [°C]	R	C	L	tmax [°C]	R	C	L
136	MATERIAL	reinforced_concrete_(stee	3.30	3	21	9	16.28	43	1	1
151	MATERIAL	insulation_0.035_W/mK	3.74	7	25	27	14.82	11	29	35
164	MATERIAL	masonry_semi-heavy	9.42	11	7	9	15.03	13	29	35
170	BC_SIMPL	exterior	1.29	3	25	27	4.11	3	1	35
174	BC_SIMPL	interior_(normal)	10.93	23	1	35	15.36	43	7	9
180	MATERIAL	masonry_heavy	1.29	3	25	27	7.66	7	7	9

Col.	Type	Name	ta [°C]	Flow in [W]	Flow out [W]
170	BC_SIMPL	exterior		0.00	43.65
174	BC_SIMPL	interior_(normal)		0.00	35.15

The temperature in the corner nodes for a specific time step are shown using the command *View → Corner Nodes*.

VOLTRA - Temperatures in corner nodes

VOLTRA data file: PilotexVoltra.vtr
Number of nodes = 13902

Time step 0002:16:00:00

R	C	L	t [°C]
3	3	3	7.09
3	3	45	7.50
3	57	3	7.08
3	57	45	7.73
27	27	19	8.69
27	27	45	13.07
27	57	19	10.90
27	57	45	16.05
55	3	3	7.08
55	3	45	7.73
55	27	19	10.90
55	27	45	16.06
55	57	3	7.25
55	57	19	14.91

The temperature in the node BC's of type POWER for a specific time step are shown using the command *View → Node BCs*.

Hint

To get the temperature in a specific grid point, add a dummy node BC with fixed power = 0 W.

F.3.3 Report output

For the items defined in the **Report Definition window** (cf. section B.24) a report can be output with records at a given frequency of time steps (cf. section B.25). The command *View → Report Output* shows the contents of the report written during the calculation. This report is automatically written to both a .txt file and .csv file in the VOLTRA data file directory.

In the report, per report time step one line of values is written (no value ('-') is given for logically impossible report items). The report values are separated with tab characters in the .txt file. Each column in the table corresponds to a report item. The meaning of the columns is explained in the header.

```
VOLTRA - Report Output

VOLTRA data file: PilotexVoltra.vtr

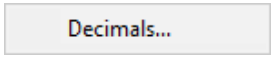
Column 1: Time [dddd:hh:mm:ss]
Column 2: Colour 1, BC temperature [°C]
Column 3: Colour 2, BC temperature [°C]
Column 4: Output node 1, temperature [°C]
Column 5: Output node 2, temperature [°C]
Column 6: Output node 3, temperature [°C]
Column 7: Output node 4, temperature [°C]
Column 8: Colour 1, sum of heat flows into object [W]
Column 9: Colour 1, sum of heat flows out of object [W]
Column 10: Colour 2, sum of heat flows into object [W]
Column 11: Colour 2, sum of heat flows out of object [W]

0000:00:00:00   -9.66   10.00   -0.79    2.11    7.99   -9.26    0.26   57.64   57.37    0.00
0000:01:00:00  -10.00   10.00   -0.79    2.11    7.99   -9.45    0.29   69.51   57.37    0.00
0000:02:00:00   -9.66   10.00   -0.80    2.11    7.99   -9.36    0.35   50.12   57.38    0.00
0000:03:00:00   -8.66   10.00   -0.81    2.11    7.99   -8.83   12.40   24.76   57.40    0.00
0000:04:00:00   -7.07   10.00   -0.80    2.11    7.99   -7.84   49.01   13.33   57.38    0.00
0000:05:00:00   -5.00   10.00   -0.76    2.14    7.99   -6.42   93.55    5.66   57.29    0.00
0000:06:00:00   -2.59   10.00   -0.66    2.21    8.00   -4.62  142.43    3.95   57.06    0.00
0000:07:00:00    0.00   20.00   -0.05    3.05    9.96   -2.55  184.90    2.48  184.52    0.00
0000:08:00:00    2.59   20.00    0.58    3.80   11.32   -0.33  216.85    1.49  168.39    0.00
0000:09:00:00    5.00   20.00    1.20    4.41   12.01    1.89  235.03    0.90  159.15    0.00
0000:10:00:00    7.07   20.00    1.90    5.04   12.54    3.98  237.44    0.59  151.36    0.00
0000:11:00:00    8.66   20.00    2.67    5.71   13.00    5.80  223.36    0.49  144.10    0.00
0000:12:00:00    9.66   20.00    3.49    6.40   13.43    7.23  193.31    0.44  137.10    0.00
0000:13:00:00   10.00   20.00    4.35    7.11   13.83    8.17  149.00    0.40  130.29    0.00
0000:14:00:00    9.66   20.00    5.19    7.80   14.21    8.57   93.33    0.47  123.68    0.00
0000:15:00:00    8.66   20.00    6.00    8.46   14.58    8.39   30.36    1.12  117.33    0.00
0000:16:00:00    7.07   20.00    6.74    9.07   14.92    7.66    0.24   38.34  111.30    0.00
0000:17:00:00    5.00   20.00    7.38    9.61   15.25    6.41    0.14  104.91  105.68    0.00
0000:18:00:00    2.59   20.00    7.89   10.06   15.54    4.75    0.29  166.66  100.53    0.00
0000:19:00:00    0.00   20.00    8.26   10.41   15.82    2.78    0.43  219.28   95.94    0.00
0000:20:00:00   -2.59   10.00    8.04    9.93   14.11    0.64    0.55  259.30    0.50   36.47
0000:21:00:00   -5.00   10.00    7.76    9.51   12.97   -1.53    0.64  283.90    1.10   25.01
0000:22:00:00   -7.07   10.00    7.45    9.19   12.49   -3.58    0.70  291.41    1.68   20.08
0000:23:00:00   -8.66   10.00    7.04    8.83   12.15   -5.36    0.73  281.38    2.43   16.51
```

0001:00:00:00	-9.66	10.00	6.54	8.42	11.86	-6.76	0.71	254.61	3.43	13.46
0001:01:00:00	-10.00	10.00	5.99	7.97	11.59	-7.68	0.65	213.03	4.66	10.71
0001:02:00:00	-9.66	10.00	5.41	7.51	11.33	-8.06	0.57	159.54	6.13	8.22
0001:03:00:00	-8.66	10.00	4.84	7.06	11.08	-7.87	0.52	97.88	7.80	6.00
0001:04:00:00	-7.07	10.00	4.32	6.64	10.84	-7.13	8.16	39.94	9.65	4.12
0001:05:00:00	-5.00	10.00	3.89	6.28	10.62	-5.88	55.12	21.77	11.57	2.54
0001:06:00:00	-2.59	10.00	3.56	6.00	10.41	-4.21	102.51	8.93	13.56	1.36
0001:07:00:00	-0.00	20.00	3.79	6.52	12.17	-2.23	149.64	4.84	142.87	0.00
0001:08:00:00	2.59	20.00	4.07	6.97	13.35	-0.08	186.62	3.26	129.74	0.00
0001:09:00:00	5.00	20.00	4.38	7.31	13.87	2.09	208.70	2.17	123.31	0.00
0001:10:00:00	7.07	20.00	4.80	7.70	14.24	4.14	214.19	1.44	118.15	0.00
0001:11:00:00	8.66	20.00	5.31	8.14	14.56	5.93	202.64	1.03	113.32	0.00
0001:12:00:00	9.66	20.00	5.91	8.64	14.86	7.34	174.80	0.85	108.59	0.00
0001:13:00:00	10.00	20.00	6.56	9.16	15.14	8.27	132.52	0.88	103.88	0.00
0001:14:00:00	9.66	20.00	7.22	9.69	15.41	8.65	78.74	1.15	99.23	0.00
0001:15:00:00	8.66	20.00	7.86	10.20	15.68	8.46	17.61	2.19	94.68	0.00
0001:16:00:00	7.07	20.00	8.45	10.67	15.93	7.72	0.06	50.73	90.33	0.00
0001:17:00:00	5.00	20.00	8.94	11.08	16.16	6.47	0.20	116.43	86.26	0.00
0001:18:00:00	2.59	20.00	9.33	11.41	16.38	4.80	0.35	177.19	82.55	0.00
0001:19:00:00	0.00	20.00	9.58	11.65	16.58	2.82	0.48	228.93	79.28	0.00
0001:20:00:00	-2.59	10.00	9.26	11.08	14.81	0.68	0.60	268.15	0.02	51.41
0001:21:00:00	-5.00	10.00	8.88	10.57	13.62	-1.49	0.69	292.03	0.10	38.28
0001:22:00:00	-7.07	10.00	8.48	10.17	13.08	-3.54	0.74	298.88	0.24	31.88
0001:23:00:00	-8.66	10.00	7.99	9.73	12.69	-5.33	0.76	288.26	0.57	26.91
0002:00:00:00	-9.66	10.00	7.42	9.24	12.35	-6.73	0.74	260.95	1.21	22.59
0002:01:00:00	-10.00	10.00	6.79	8.73	12.04	-7.66	0.69	218.87	2.12	18.68
0002:02:00:00	-9.66	10.00	6.15	8.21	11.74	-8.04	0.60	164.92	3.32	15.14
0002:03:00:00	-8.66	10.00	5.52	7.71	11.46	-7.85	0.53	102.84	4.70	11.92
0002:04:00:00	-7.07	10.00	4.95	7.25	11.19	-7.11	6.96	43.29	6.26	9.08
0002:05:00:00	-5.00	10.00	4.47	6.84	10.94	-5.86	53.42	24.28	7.92	6.63
0002:06:00:00	-2.59	10.00	4.10	6.51	10.70	-4.19	100.38	10.69	9.56	4.54
0002:07:00:00	-0.00	20.00	4.29	6.99	12.44	-2.21	146.50	5.29	136.22	0.00
0002:08:00:00	2.59	20.00	4.53	7.41	13.59	-0.07	183.67	3.63	123.58	0.00
0002:09:00:00	5.00	20.00	4.80	7.72	14.10	2.10	205.93	2.46	117.60	0.00
0002:10:00:00	7.07	20.00	5.19	8.07	14.45	4.16	211.59	1.68	112.85	0.00
0002:11:00:00	8.66	20.00	5.67	8.49	14.75	5.94	200.21	1.21	108.42	0.00
0002:12:00:00	9.66	20.00	6.25	8.96	15.03	7.35	172.53	1.01	104.04	0.00
0002:13:00:00	10.00	20.00	6.87	9.46	15.30	8.28	130.44	1.03	99.67	0.00
0002:14:00:00	9.66	20.00	7.51	9.96	15.56	8.66	76.84	1.32	95.32	0.00
0002:15:00:00	8.66	20.00	8.13	10.45	15.81	8.47	15.92	2.42	91.06	0.00
0002:16:00:00	7.07	20.00	8.69	10.90	16.05	7.73	0.06	52.51	86.97	0.00
0002:17:00:00	5.00	20.00	9.17	11.30	16.28	6.48	0.21	118.08	83.14	0.00
0002:18:00:00	2.59	20.00	9.54	11.61	16.49	4.81	0.35	178.72	79.66	0.00
0002:19:00:00	0.00	20.00	9.77	11.84	16.68	2.83	0.49	230.35	76.60	0.00
0002:20:00:00	-2.59	10.00	9.43	11.25	14.90	0.68	0.60	269.46	0.01	53.88
0002:21:00:00	-5.00	10.00	9.04	10.73	13.70	-1.49	0.69	293.24	0.06	40.55
0002:22:00:00	-7.07	10.00	8.63	10.32	13.15	-3.54	0.75	300.01	0.17	33.94
0002:23:00:00	-8.66	10.00	8.13	9.87	12.76	-5.33	0.77	289.30	0.43	28.74
0003:00:00:00	-9.66	10.00	7.55	9.37	12.41	-6.73	0.75	261.91	0.99	24.20

F.4 Settings

Settings submenu



F.4.1 Output precision

Menu command: *Settings → Decimals...*



CHAPTER G VOLTRA TEMPLATE FILES

G.1 General

VOLTRA makes use of template files containing default information for a number of functions. These template files are each saved in the application data folder or personal folder (see A.6) of the user account on the computer where VOLTRA is installed. The path to the application data folder is C:/Users/*username*/Appdata/Roaming/Physibel/VOLTRA9 (AppData is a hidden folder by default in Windows).

The template files used by VOLTRA are:

- ColourDatabase.vtr (G.2)

The use of these template files in different VOLTRA functions has been outlined throughout the manual. In the following sections, handling and customization of these template files is briefly discussed.

G.2 Colour database

The colour database file ColourDatabase.vtr is saved in the application data folder (C:/Users/*username*/Appdata/Roaming/Physibel/VOLTRA9). The colour database contains predefined material and boundary condition properties and is loaded at every VOLTRA start-up (see B.1).

It is advisable to add frequently used materials and boundary conditions to the colour database.

These predefined colours of the colour database are then available for new colours in any document.

The command *File → Open Colour Database...* allows to modify the 256 available predefined colours of the colour database. After editing the colour data, the file should be saved (*File → Save*) to adapt the colour database on disk.

Alternatively, if a user wants to save a custom file as colour database, this is possible by saving it with the name ColourDatabase.vtr in the application data folder (*File → Save as...*) thus overwriting the default file. In the application data folder, the default colour database is present as ColourDatabase_PhysibelDefault.vtr. This file can be saved as ColourDatabase.vtr to restore the original colour database.

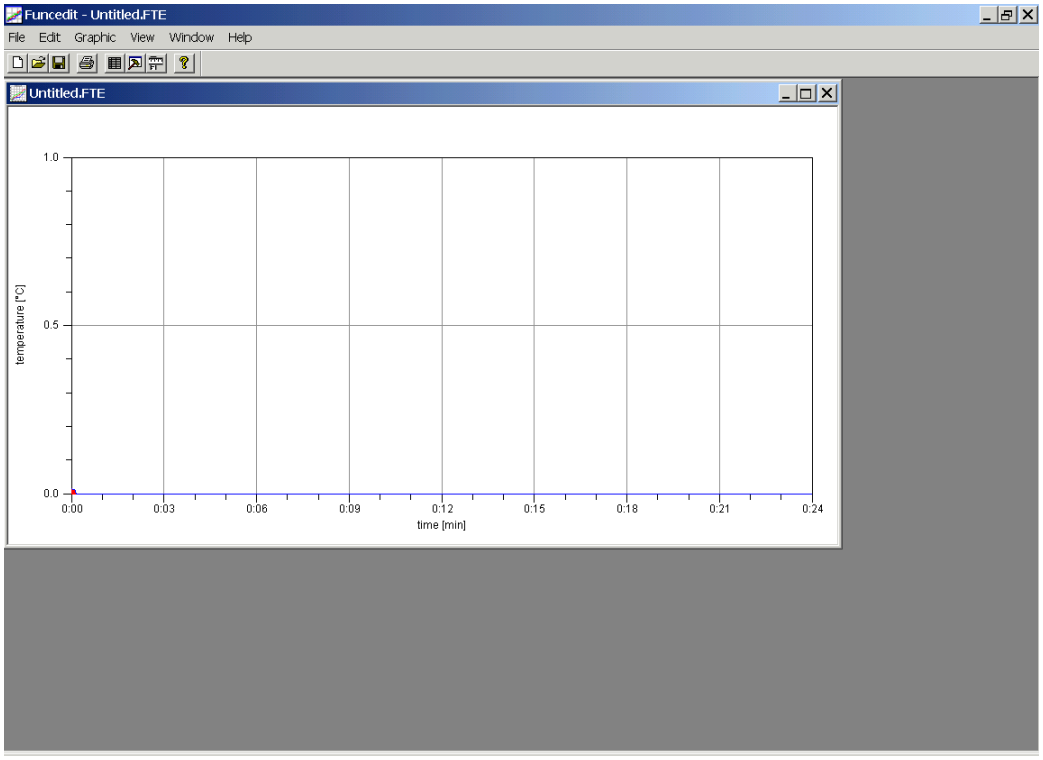
The ColourDatabase.vtr file replaces the VOLTRA.vtr file in older versions of VOLTRA. Users who had been working with a customised VOLTRA.vtr file, can thus easily transfer to the new system by opening VOLTRA.vtr and saving it with the name ColourDatabase.vtr in the application data folder.

CHAPTER H PROGRAM FUNCEDIT

H.1 Introduction

The program FUNCEDIT is a utility program to edit and visualise external function files for use in non-steady state Physibel programs (BISTRA, VOLTRA), where they are used as dynamic boundary condition specifications (see B.12.2).

After starting FUNCEDIT or creating a new function using the command *File → New*, a null temperature function of length 24 hours (with hourly values of 0°C) is displayed.



An existing function file on disk is read using the command *File → Open...*. The function type can be chosen in the Open dialog box.

H.2 Function type

The following function types are available:

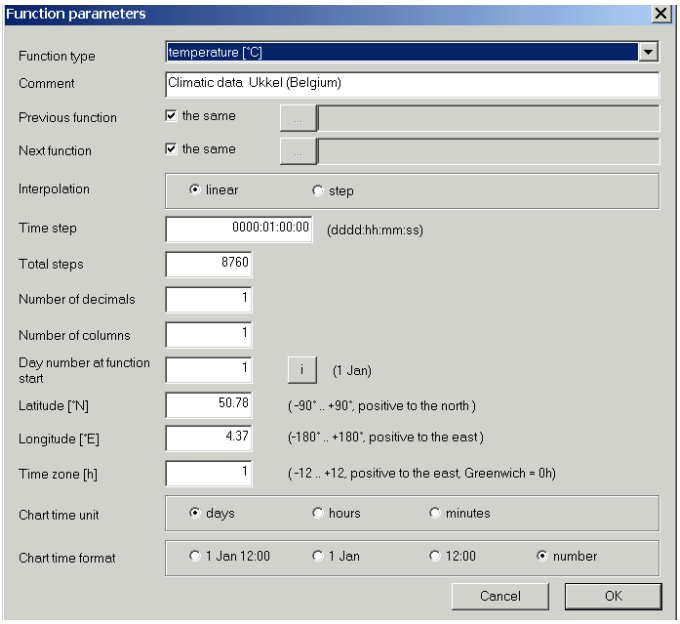
Function type	*Unit*	*ext.*	*Use in VOLTRA*
temperature	[°C]	.fte	boundary condition temperature
horizontal global solar radiation	[W/m^2]	.fsg	boundary condition solar zone
horizontal diffuse solar radiation	[W/m^2]	.fsd	boundary condition solar zone
direct solar radiation	[W/m^2]	.fsb	not used
solar radiation	[W/m^2]	.fse	not used
infrared radiation	[W/m^2]	.fir	boundary condition heat flux
power	[W]	.ffh	boundary condition power
thermal conductivity	[W/(m.K)]	.fco	material property time-dependent thermal conductivity
surface heat transfer coefficient	[W/(m^2.K)]	.fht	boundary condition total surface heat transfer coefficient or convective heat transfer coefficient
ventilation rate	[/h]	.fvr	ventilation volume flow rate [m^3/s]

Remark

The ventilation rate in FUNCEDIT is shown with unit /h, which is used in the (now deprecated) Physibel program CAPSOL. However in VOLTRA the same function type is used for ventilation volume flow rates, with unit m^3/s. So for VOLTRA ventilation functions the unit specifier [/h] in FUNCEDIT should be read as [m^3/s].

H.3 Function parameters

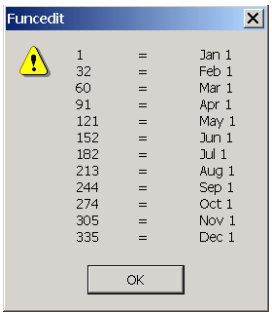
General header information of a function file is edited via the command *Edit → Parameters...*.



A dialog box is opened where the following parameters can be edited:

- Function type: see section H.2.
- Comment: text description of the function.
- Previous function and next function: file name of a function that precedes the current function (previous function) or comes next to the current function (next function). The previous and/or next function may be the same as the current function, in which case the check box “the same” is marked.
  If the time span of a VOLTRA simulation is larger than the time span of the function definition, it is necessary to know which function comes next to the current one. In case a start-up calculation is used, a previous function is required. If the previous and the next function are checked as the same, then the current function corresponds to the definition of an infinite periodic function, with period the length of the current function.
- Interpolation: linear or step. Intermediate function values are either obtained by linear interpolation (linear function) or remain constant during the whole time step interval (step function).
- Time step [dddd:hh:mm:ss]: time interval between two successive function values in the function data window (see section H.4).
- Total steps: number of time steps required to span the total function length. This number is equal to the number of function values minus 1. The function length or duration equals the time step multiplied by the total number of steps.
- Number of decimals: number of digits after the decimal point, for all function values in the function data window (see section H.4) and in the function file.
- Number of columns: used in the function data window (see section H.4) and in the function file.

- Day number at function start [1..365]: day number in the year on which the function starts. The "i" button gives information about the start days of each month.



- Latitude [°N], longitude [°E] and time zone [h]: to position a function climate station on the globe.
- Chart time unit and chart time format: labelling information for the time axis in the function chart. This information is not stored in the function file.

H.4 Function data

The function values at regular time step intervals can be edited in the function data window, opened with the command *Edit → Data*. The table format is controlled by the function parameters (see section H.3).

In the left (grey) column a time indication of the first value in the row is given. The table can be navigated by the mouse or by the arrow keys.

A new value can be inserted after the current one using the command *Edit → Insert After* or the Insert key. The "total steps" parameter in the function parameters dialog box is adapted accordingly.

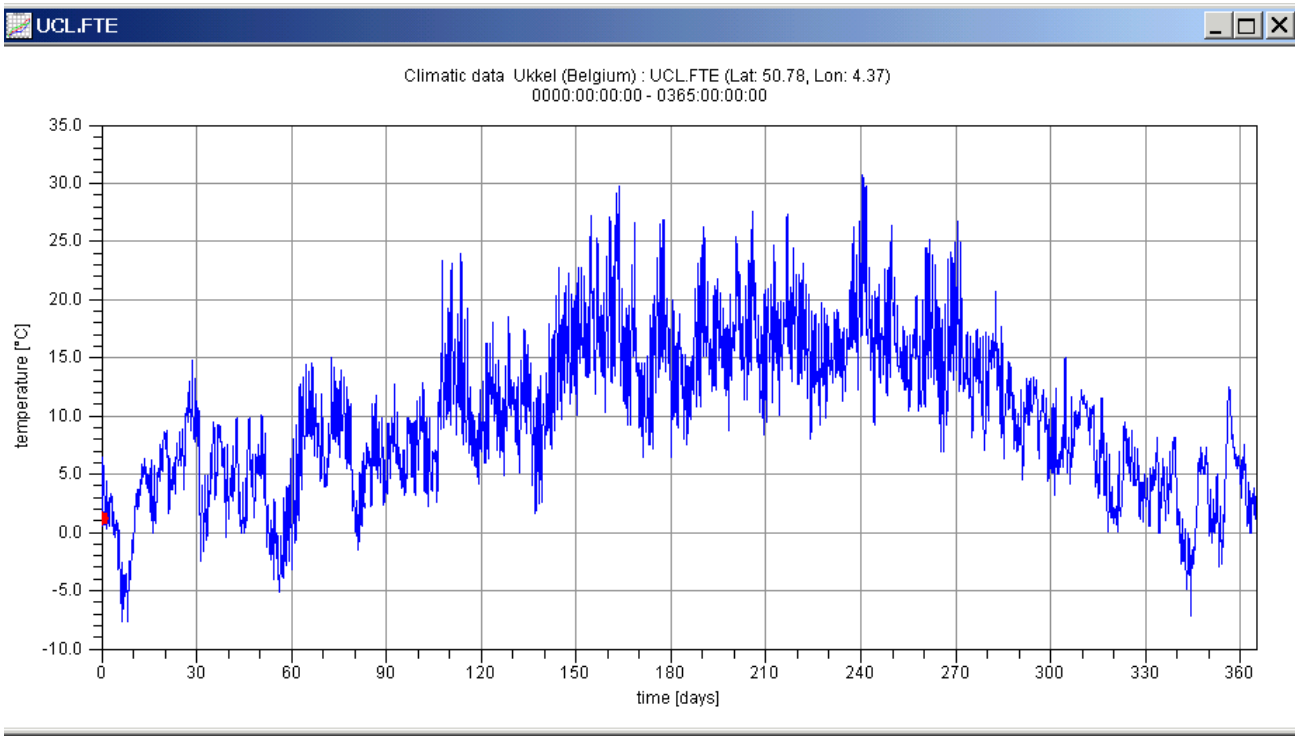
A value can be deleted using the command *Edit → Delete Value* or the Delete key.

Individual table cells can be copied and pasted by using the commands *Edit → Copy* (or <Ctrl+C>) and *Edit → Paste* (or <Ctrl+V>).

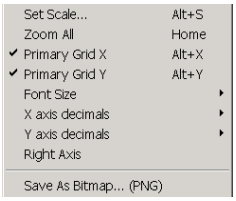
H.5 Graphic parameters

The window that is opened when a new function is created or an existing function file is read, contains a graphic representation of the numerical function data entered in the data window.

The red point in the graph corresponds to the function value in the current cell of the data window.

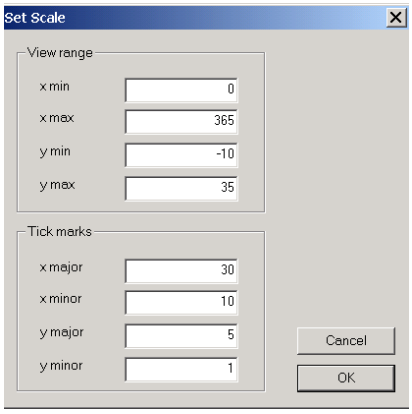


The parameters about the graph can be edited in the Graphic submenu.



The command *Graphic → Set Scale...* opens a dialog box containing scale parameters.

The view range for both axes and tick mark distances can be set.



When a rectangle is drawn in the graph with mouse dragging (holding down the left mouse button), a zoom rectangle is defined. The corresponding zoom view is drawn when the left mouse button is released. The initial total view (with view range as defined in the Set Scale dialog box) can be restored by using the command *Graphic → Zoom All* or pressing the Home key.

The commands *Graphic → Primary Grid X* and *Graphic → Primary Grid Y* enable to show grid lines through the primary tick marks on both axes X and Y.

The command *Graphic → Right Axis* allows to draw also a vertical axis with tick marks at the right side of the chart border.

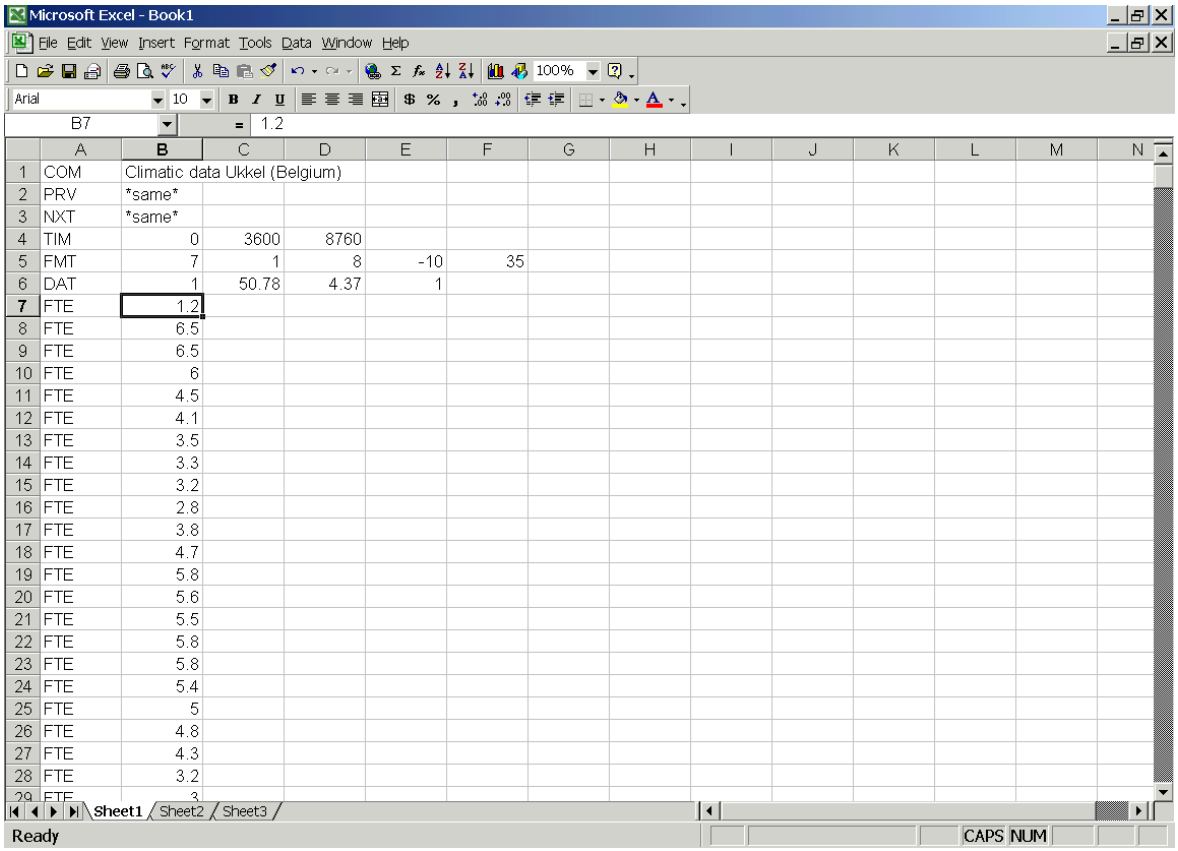
The command *Graphic → Save As Bitmap (PNG)* opens a dialog box where you can enter the file name and size of the .png file, which will be saved in the same directory as where the function is stored.

H.6 Using a spreadsheet program to create functions

A function file can also be created using Microsoft Excel or another spreadsheet program.

The spreadsheet contains 6 header rows (starting with the keywords COM, PRV, NXT, TIM, FMT and DAT) followed by data rows with function values. The syntax of the input data is given in the table below. Save the file (as text) in the function files directory, with the extension corresponding to the function type (cf. section H.2).

If an existing function file, created with FUNCEDIT, has to be modified using a spreadsheet program, it is recommended first to set the parameter "number of columns" in the Function Parameters dialog box equal to 1. Then there will be only 1 column with function values in the spreadsheet (as in the figure below).



	A	B	C	D	E	F
1	COM	Climatic data Ukkel (Belgium)				
2	PRV	*same*				
3	NXT	*same*				
4	TIM	0	3600	8760		
5	FMT	7	1	8	-10	35
6	DAT	1	50.78	4.37	1	
7	FTE	1.2				
8	FTE	6.5				
9	FTE	6.5				
10	FTE	6				
11	FTE	4.5				
12	FTE	4.1				
13	FTE	3.5				
14	FTE	3.3				
15	FTE	3.2				
16	FTE	2.8				
17	FTE	3.8				
18	FTE	4.7				
19	FTE	5.8				
20	FTE	5.6				
21	FTE	5.5				
22	FTE	5.8				
23	FTE	5.8				
24	FTE	5.4				
25	FTE	5				
26	FTE	4.8				
27	FTE	4.3				
28	FTE	3.2				
29	FTE	3				

row number	code column A	input data				
		column B	column C	column D	column E	column F
1	COM	comment				
2	PRV	name of the previous function *				
3	NXT	name of the next function *				
4	TIM	interpolation: 0 = linear / 1 = step	time step (in seconds)	total number of steps		
5	FMT	total number of characters of the function values	number of digits after decimal point	number of columns of function values	minimum ordinate	maximum ordinate
6	DAT	date number at function start	latitude	longitude	time zone	
7 - ... - (7 + total steps)	function type**	function value	(function value)	(function value)	(function value)	(function value)

* The extension of the file name may be omitted; Omit this line to specify the same previous / next function (or use the name *same*).

** The function type as described in section H.2.

CHAPTER I RADCON MODULE

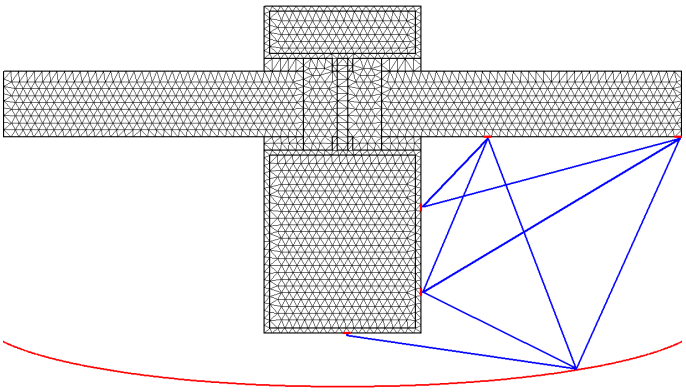
I.1 Purpose

The RADCON module for VOLTRA is a program add-on feature to simulate more realistically the heat transfer between material surfaces and the adjacent environments.
The heat transfer has the following components:

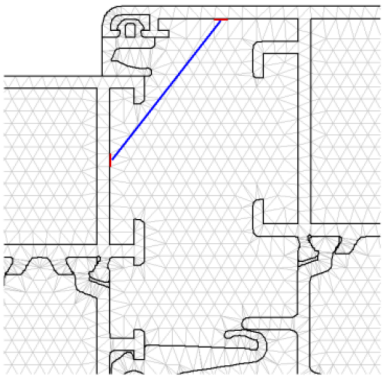
- Radiation: based on view factors and material surface emissivities.
- Convection: based on a (calculated) convective heat transfer coefficient applied to the contact surface between material and environment.
- Conduction: in a (transparent) material with given thermal conductivity.

The RADCON module offers a physically more correct alternative for:

- The global surface heat transfer coefficient.
- The equivalent thermal conductivity of an air cavity.



principle of view factor based radiation heat transfer using BC_SKY (at a surface)



principle of view factor based radiation heat transfer using TRANSMAT (in a cavity)

I.2 Implementation

The following colour types require the use of the RADCON module in VOLTRA:

- TRANSMAT for a transparent material with known (or calculated) thermal conductivity (normally of still air or another gas) according to EN ISO 10077-2)
  Additionally to the gas conduction there is radiative heat transfer between the adjacent material surfaces (using their emissivities).
  If the transparent material is not fully surrounded by material surfaces, then the open side at the bitmap border is considered as an adiabatic surface, i.e. a radiation mirror with emissivity $\varepsilon = 0$.
  If there is convection within the cavity, the thermal conductivity for still gas should be multiplied by the Nusselt number.
- BC_FREE for an enclosure (normally fully surrounded by material surfaces, e.g. a cavity or a room) with unknown air temperature (free-floating air temperature), known surface heat flux (positive or negative), known (or calculated) convective heat transfer coefficient and known injected convective heat power (positive or negative).

The unknown air temperature is calculated from the convective heat balance equation: Σ (convective heat transfer from environment to surface) = injected convective heat power. The radiation between the different surfaces bordering the environment are calculated based on view factors and surface emissivities.
When the environment is not completely surrounded by material surfaces, the open end (at the bitmap border) is considered to be a symmetry plane for heat transfer. The symmetry plane is simulated as a radiation mirror (with emissivity ε = 0).

- BC_SKY for an environment (normally not fully surrounded by material surfaces, i.e. open to the sky) with known air temperature, known radiation temperature, known surface heat flux (positive or negative) and known (or calculated) convective heat transfer coefficient. This implies there can be a resultant radiative heat flow and a resultant convective heat flow from the environment to the material surfaces.
- BC_NOSKY for an environment with known air temperature, known surface heat flux (positive or negative) and known (or calculated) convective heat transfer coefficient. There is no radiation exchange with the environment (i.e. no sky radiation). The surfaces bordering the environment exchange radiation based on view factors and surface emissivities.
  The known air temperature implies there can be a resultant convective heat flow from the environment to the material surfaces.

For use with RADCON the colour types MATERIAL and EQUIMAT also have a surface emissivity (defined in the **Colours window**), which is applied to the radiative heat transfer for the RADCON boundary conditions bordering the material.

Combined convective and radiative heat transfer (BC_SIMPL, BC_FRE_S)

The global (combined convective and radiative) heat flow is:

$$Q = h \cdot (\theta_s - \theta_{bc}) \cdot A$$

Q = global heat flow [W]
h = global heat transfer coefficient [W/(m^2.K)]
θ_s = surface temperature [°C]
θ_{bc} = environmental temperature [°C]
A = surface area [m^2]

In this formula the environmental temperature is not defined precisely. It can be the air temperature, the dry resultant temperature or an interpolated value. Depending on that choice, an appropriate value for the global heat transfer coefficient must be chosen.

Convective heat transfer (BC_FREE, BC_SKY, BC_NOSKY)

The convective heat flow is:

$$Q_c = h_c \cdot (\theta_s - \theta_a) \cdot A$$

Q_c = convective heat flow [W]
h_c = convective heat transfer coefficient [W/(m^2.K)]
θ_s = surface temperature [°C]
θ_a = air temperature [°C]
A = surface area [m^2]

The convective heat transfer coefficient h_c can be calculated based on the given subtype.

Black radiation (BC_FREE, BC_SKY, BC_NOSKY, TRANSMAT)

The radiative heat flow between two black surfaces is:

$$Q_{ij} = A_i \cdot F_{ij} \cdot h_{rb} \cdot (\theta_{sbi} - \theta_{sbj})$$

Q_{ij} = radiative heat flow between black surface i and black surface j [W]
A_i = area of surface i [m²]
F_{ij} = view factor from surface i to surface j (between 0 and 1)
h_{rb} = black radiation heat transfer coefficient [W/(m².K)]
θ_{sbi} = black surface temperature of surface i [°C]
θ_{sbj} = black surface temperature of surface j [°C]

For linear radiation the black radiation heat transfer coefficient is constant, defined as a calculation parameter (default value 5.1 W/(m².K)). The value depends on the temperature range of the problem considered. Section I.3 contains a table with values of h_{rb}, depending on the mean temperature θ_m [°C] of the surfaces i and j and the temperature difference between the surfaces $\Delta\theta_{ij}$ [°C]. The table shows that the h_{rb}-value mainly depends on the mean surface temperature θ_m.

The default value 5.1 W/(m².K) is the h_{rb}-value at 10°C.

For non-linear radiation:

$$h_{rb} = \sigma \cdot (T_{sbi}^2 + T_{sbj}^2) \cdot (T_{sbi} + T_{sbj})$$

σ = 5.67 10^{-8} W(m².K⁴) (Stefan-Boltzmann constant)
T_{sbi} = θ_{sbi} + 273.16 [K] (absolute temperature of black surface i)
T_{sbj} = θ_{sbj} + 273.16 [K] (absolute temperature of black surface j)

The black radiation heat transfer coefficients are recalculated at the start of each new iteration cycle based on the temperatures obtained in the last iteration cycle.

Grey radiation (BC_FREE, BC_SKY, BC_NOSKY, TRANSMAT)

Real surfaces are not black (emissivity ε = 1) but grey (emissivity ε < 1).

In the RADCON module each grey surface is connected to a black surface node. All black surface nodes are connected to each other in a star diagram.

The radiative heat flow between the grey surface with given emissivity and the corresponding black surface is:

$$Q_r = A \cdot \frac{\varepsilon}{1-\varepsilon} \cdot h_{rb} \cdot (\theta_{sb} - \theta_s)$$

Q_r = radiative heat flow between grey surface and black surface [W]
A = surface area [m²]
ε = surface emissivity (0 ≤ ε < 1)
h_{rb} = black radiation heat transfer coefficient [W/(m².K)]
θ_{sb} = black surface temperature [°C]
θ_s = (real) surface temperature [°C]

For non-linear radiation:

$$h_{rb} = \sigma \cdot (T_{sb}^2 + T_s^2) \cdot (T_{sb} + T_s)$$

T_{sb} = θ_{sb} + 273.16 [K]
T_s = θ_s + 273.16 [K]

The black radiation heat transfer coefficients are recalculated at the start of each new iteration cycle based on the temperatures obtained in the last iteration cycle.

Iteration cycles for non-linear radiation (BC_FREE, BC_SKY, BC_NOSKY, TRANSMAT)

Iteration cycles are used to calculate non-linear radiative heat transfer. At the start of each iteration cycle the black radiation heat transfer coefficients are updated using the local temperatures available at that moment.
For the first iteration cycle the black radiation heat transfer coefficients have a constant value as defined in the **Calculation Parameters dialog box**.
The total number of iteration cycles is defined in the **Calculation Parameters dialog box**.
Normally 5 iteration cycles are sufficient. A low number of iteration steps in the last iteration cycle ensures that the number of iteration cycles is OK.

View factors (BC_FREE, BC_SKY, BC_NOSKY, TRANSMAT)

The view factors are calculated for all elementary surfaces (corresponding to edges in the mesh triangulation) bordering the environment (type BC_FREE, BC_SKY, BC_NOSKY or TRANSMAT).

Several environments of the same type (e.g. with different convective heat transfer coefficients) may touch one another. The view factors are calculated for the union space of these adjacent environments.
Adjacent environments of different type are not allowed. Then an error message is displayed at the start of the system calculation (*Calc → Calc System*).
A boundary condition of type BC_FREE, BC_SKY, BC_NOSKY or TRANSMAT is not allowed to occur in different zones (separated by a material). Each zone must be considered as a separate environment (having a different colour). An error message is displayed at the start of the system calculation, when this condition is not satisfied.

First the view factor between two (elementary) surfaces, not obstructed by other surfaces, is calculated using an exact analytic formula. A very small view factor (i.e. smaller than the calculation parameter *smallest accepted view factor*) is neglected (rounded to 0). Then possible obstruction by other surfaces is estimated using several test rays between random points of both surfaces. The number of test rays is proportional to the unobstructed view factor. The maximum number of test rays (i.e. when the unobstructed view factor = 1) is defined as the calculation parameter *number of visibility rays between radiative surfaces*).
After calculation of all view factors a correction is made to smooth away possible errors (e.g. due to the limited number of visibility test rays), so that the sum of view factors always equals 1, and the view factor reciprocities are respected.

The total number of calculated view factors (and hence the calculation time and required memory space) is a quadratic function of the number of elementary surfaces (= triangulation edges) in the largest view factor zone.

I.3 Table for black radiation heat transfer coefficient

The following table shows the black radiation heat transfer coefficient.
The values are obtained using the formula:

$$h_{rb} = \sigma \cdot (T_i^2 + T_j^2) \cdot (T_i + T_j)$$

with σ = 5.67*10^{-8} W/(m^2.K^4) (Stefan-Boltzmann constant)

black radiation heat transfer coeff. h_{rb} [W/m^{2}K]		temperature difference between the surfaces i and j $\Delta\theta_{ij}$[°C]										
		0.00	5.00	10.00	15.00	20.00	25.00	30.00	35.00	40.00	45.00	50.00
mean temp. θ_m[°C]	–20.00	3.68	3.68	3.68	3.68	3.69	3.69	3.69	3.70	3.70	3.71	3.72
	–15.00	3.90	3.90	3.90	3.91	3.91	3.91	3.92	3.92	3.93	3.93	3.94
	–10.00	4.13	4.13	4.13	4.14	4.14	4.14	4.15	4.15	4.16	4.16	4.17
	–5.00	4.37	4.37	4.37	4.38	4.38	4.38	4.39	4.39	4.40	4.40	4.41
	0.00	4.62	4.62	4.62	4.63	4.63	4.63	4.64	4.64	4.65	4.65	4.66
	5.00	4.88	4.88	4.88	4.88	4.89	4.89	4.90	4.90	4.91	4.91	4.92
	10.00	5.15	5.15	5.15	5.15	5.16	5.16	5.16	5.17	5.17	5.18	5.19
	15.00	5.43	5.43	5.43	5.43	5.43	5.44	5.44	5.45	5.45	5.46	5.47
	20.00	5.71	5.71	5.72	5.72	5.72	5.72	5.73	5.73	5.74	5.75	5.76
	25.00	6.01	6.01	6.01	6.02	6.02	6.02	6.03	6.03	6.04	6.05	6.05
	30.00	6.32	6.32	6.32	6.32	6.33	6.33	6.33	6.34	6.35	6.35	6.36
	35.00	6.64	6.64	6.64	6.64	6.64	6.65	6.65	6.66	6.66	6.67	6.68
	40.00	6.96	6.97	6.97	6.97	6.97	6.98	6.98	6.99	6.99	7.00	7.01
	45.00	7.30	7.30	7.31	7.31	7.31	7.32	7.32	7.33	7.33	7.34	7.35
	50.00	7.65	7.65	7.66	7.66	7.66	7.67	7.67	7.68	7.68	7.69	7.70

I.4 Table for emissivities

The table below contains values of the long-wave emissivity ε of some building materials taken from:

- H.C. Hottel, Heat Transmission, 1954

- J.S. Cammerer, Wärme- und Kälteschutz in der Industrie, 1962

- Stichting Bouwresearch, Eigenschappen van bouw- en isolatiematerialen, rapport 9, 1974

building material	long-wave emissivity at T = 300K ε [-]
black surface	1
aluminium, polished	< 0.07
aluminium, oxidised	0.11 à 0.3
brass, polished	0.038
copper, smoothed	0.09
copper, oxidised	0.78 - 0.79
cast iron	0.44
iron, oxidised and rusted red	0.61
lead, gray oxidised	0.28
tin	0.043 - 0.064
zinc, galvanised	0.23
zinc, oxidised	> 0.11
asbestos board	0.96
bitumen	0.92
brick, red, rough but no gross irregularities	0.92 - 0.93
concrete	0.88
glass	0.94
cement-tiles	> 0.85
roofing-tiles, red	0.85
roofing paper	0.91
rubber, hard and glossy	0.94
wood	0.86
aluminium painting	0.42
enamel, white	>0.90
metal paint, black	0.88
water	0.95 - 0.96
other materials (approximation) white, yellow, green, red, blue, grey, black	0.85 à 0.95

The table below contains values of the long-wave emissivity ε from EN ISO 10077-2:2017:

description	ε [–]
Untreated aluminium surfaces	0.1
Slightly oxidized aluminium surface (up to 5 µm)	0.3
Metallic surfaces (general, including galvanized)	0.3
Anodized, painted or powder coated surfaces	0.9

I.5 Tables for convective heat transfer coefficient

The convective heat flow density is calculated using the following formula:

$q_c = h_c \cdot (\theta_a - \theta_s)$ with: q_c= convective heat flow density [W/m^2]

h_c = convective heat transfer coefficient [W/(m^2.K)]

θ_a= air temperature [°C]

θ_s = surface temperature [°C]

Table 1 lists values of h_c for air cavities according to EN ISO 10077-2, depending on the temperature difference $\Delta\theta$ over the cavity and the depth *d* of the cavity. Above the bold line, the h_c-value depends only on the cavity depth (same h_c-values on each row), while underneath the bold line the h_c-value depends only on the temperature difference over the cavity (same h_c-values in each column). Above the bold line there is in fact only conduction (still air). Below the line natural convection is considered.

Table 1: convective heat transfer coefficient (surface-to-surface) for air cavities (EN ISO 10077-2)

convective heat transfer coefficient h_c [W/(m².K)]		temperature difference over the cavity $\Delta\theta$ [°C]									
		0.500	1.000	1.500	2.000	3.000	4.000	5.000	6.000	8.000	10.000
depth of the cavity d [m]	0.001	50.000	50.000	50.000	50.000	50.000	50.000	50.000	50.000	50.000	50.000
	0.002	25.000	25.000	25.000	25.000	25.000	25.000	25.000	25.000	25.000	25.000
	0.003	16.667	16.667	16.667	16.667	16.667	16.667	16.667	16.667	16.667	16.667
	0.004	12.500	12.500	12.500	12.500	12.500	12.500	12.500	12.500	12.500	12.500
	0.005	10.000	10.000	10.000	10.000	10.000	10.000	10.000	10.000	10.000	10.000
	0.006	8.333	8.333	8.333	8.333	8.333	8.333	8.333	8.333	8.333	8.333
	0.007	7.143	7.143	7.143	7.143	7.143	7.143	7.143	7.143	7.143	7.143
	0.008	6.250	6.250	6.250	6.250	6.250	6.250	6.250	6.250	6.250	6.250
	0.009	5.556	5.556	5.556	5.556	5.556	5.556	5.556	5.556	5.556	5.556
	0.010	5.000	5.000	5.000	5.000	5.000	5.000	5.000	5.000	5.000	5.000
	0.011	4.545	4.545	4.545	4.545	4.545	4.545	4.545	4.545	4.545	4.545
	0.012	4.167	4.167	4.167	4.167	4.167	4.167	4.167	4.167	4.167	4.167
	0.013	3.846	3.846	3.846	3.846	3.846	3.846	3.846	3.846	3.846	3.846
	0.014	3.571	3.571	3.571	3.571	3.571	3.571	3.571	3.571	3.571	3.571
	0.015	3.333	3.333	3.333	3.333	3.333	3.333	3.333	3.333	3.333	3.333
	0.016	3.125	3.125	3.125	3.125	3.125	3.125	3.125	3.125	3.125	3.145
	0.017	2.941	2.941	2.941	2.941	2.941	2.941	2.941	2.941	2.941	3.145
	0.018	2.778	2.778	2.778	2.778	2.778	2.778	2.778	2.778	2.920	3.145
	0.019	2.632	2.632	2.632	2.632	2.632	2.632	2.632	2.653	2.920	3.145
	0.020	2.500	2.500	2.500	2.500	2.500	2.500	2.500	2.653	2.920	3.145
	0.025	2.000	2.000	2.000	2.000	2.106	2.318	2.497	2.653	2.920	3.145
	0.030	1.667	1.667	1.671	1.839	2.106	2.318	2.497	2.653	2.920	3.145
	0.035	1.429	1.460	1.671	1.839	2.106	2.318	2.497	2.653	2.920	3.145
	0.040	1.250	1.460	1.671	1.839	2.106	2.318	2.497	2.653	2.920	3.145
	0.045	1.159	1.460	1.671	1.839	2.106	2.318	2.497	2.653	2.920	3.145
	0.050	1.159	1.460	1.671	1.839	2.106	2.318	2.497	2.653	2.920	3.145
	0.060	1.159	1.460	1.671	1.839	2.106	2.318	2.497	2.653	2.920	3.145
	0.070	1.159	1.460	1.671	1.839	2.106	2.318	2.497	2.653	2.920	3.145
	0.080	1.159	1.460	1.671	1.839	2.106	2.318	2.497	2.653	2.920	3.145
	0.090	1.159	1.460	1.671	1.839	2.106	2.318	2.497	2.653	2.920	3.145
	0.100	1.159	1.460	1.671	1.839	2.106	2.318	2.497	2.653	2.920	3.145

Table 2 lists values of h_c for exterior surfaces, depending on the air velocity v and the temperature difference $\Delta\theta$ between surface and environment (from "Informatiemap voor bouwfysici", 1991).

Table 2: the convective heat transfer coefficient for exterior environments

convective heat transfer coefficient h_c [W/(m².K)]		temperature difference object -environment $\Delta\theta$[°C]			
		5	10	20	40
air velocity *v* [m/s]	0.5	7	8	9	11
	1	9	10	11	13
	2	12	13	15	16
	3	19	20	21	23
	4	25	26	27	28
	5	30	31	32	34

The following formulas define the convective heat transfer coefficient for interior environments (from "Element 29: Wärmeschutz und Energie im Hochbau", 1990). Table 3 lists the h_c-values according to these formulas.

$h_c = 1.31 \cdot \sqrt[3]{\Delta\theta}$ for vertical constructions (walls) with a horizontal heat flow

$h_c = 1.77 \cdot \sqrt[4]{\Delta\theta}$ for vertical constructions (window) with a horizontal heat flow

$h_c = 1.52 \cdot \sqrt[3]{\Delta\theta}$ for horizontal constructions (walls) with an upward heat flow

$h_c = \frac{0.59 \cdot \sqrt[4]{\Delta\theta}}{L}$ for horizontal constructions (walls) with a downward heat flow

with h_c = convective heat transfer coefficient [W/(m².K)]
$\Delta\theta$ = temperature difference object - environment [°C]
L = ½ (length x width) of the construction [m²]

Table 3: the convective heat transfer coefficient for interior environments

convective heat transfer coefficient h_c [W/(m².K)]		vertical constructions horizontal heat flow		horizontal constructions	
		walls	windows	upgoing heat flow	downgoing heat flow*
temperature difference object – environment $\Delta\theta$ [°C]	1	1.31	1.77	1.52	0.39
	2	1.65	2.10	1.92	0.47
	3	1.89	2.33	2.19	0.52
	4	2.08	2.50	2.41	0.56
	5	2.24	2.65	2.60	0.59
	6	2.38	2.77	2.76	0.62
	7	2.51	2.88	2.91	0.64
	8	2.62	2.98	3.04	0.66
	9	2.72	3.07	3.16	0.68
	10	2.82	3.15	3.27	0.70
	11	2.91	3.22	3.38	0.72
	12	3.00	3.29	3.48	0.73
	13	3.08	3.36	3.57	0.75
	14	3.16	3.42	3.66	0.76
	15	3.23	3.48	3.75	0.78

* using L = 5 m