# LORD 3.2

## LOgical R-Determination

- Modelling and identification software for thermal systems
- Based on the MRQT/PASTA package
- Developed for PASLINK EEIG, Violetstraat 21-23, B-1000 Brussels
- Written by Olaf Gutschker, BTU Cottbus / Angewandte Physik

## Introduction

LORD is a software for the modelling and calculation of thermal systems. It is a further development of the software package "MRQT/PASTA".
The thermal systems can be simple components (walls, windows etc), whole rooms or more complicated systems. For the modelling so-called RC-networks are used (analogical to electrical RC-networks). Some of the values of the individual elements of the network can be known (e.g. material parameters or from calibration measurements), the others will be calculated by parameter identification. For this calculation measured data (temperatures, heat fluxes) are needed. When all parameters of the thermal system are identified some global properties of the system can be calculated (UA- and gA-value).

## System requirements

LORD requires Microsoft Windows 95 or higher. The minimum screen resolution is 800 x 600. There are no special requirements to the used computer. However, most of the calculations are time-consuming. Therefore, using a high-performance computer is advisable.

## Installation

Copy the whole folder "LORD" anywhere on your harddisk. It contains:

"lord32.exe"        - the executable program
folder "\data"      - it contains the data files (*.dat or *.csv)
folder "\models"    - it contains model files (*.mod)
folder "\results"   - it contains the result files (*.res and *.log)

Additionally, as an example there are 3 data sets and 3 model files:

data_b01.dat    - data file from the Second System Identification Competition
data_b01.mod    - the model for this case
ref_btu.dat     - data set from PASSYS reference wall
ref_btu.mod     - the model for this case
iqbtu.dat       - data from a measurement within the IQ-TEST project
iqbtu.mod       - the model for this case

# RC-networks

The idea of modelling thermal systems by RC-networks will be illustrated by a simple example – a homogenous one-layered wall. The thermal behaviour of such a wall can be described by the law of heat conduction:

$$\frac{\delta T}{\delta t} = \alpha \cdot \frac{\delta^2 T}{\delta x^2} \qquad (\alpha \text{ ... thermal diffusivity})$$

If the boundary conditions (time-dependent surface temperatures on both sides of the wall) and the initial conditions (temperature field within the wall at a certain time) are known, the time-dependent temperature field within the wall can be calculated using this equation. This temperature field is continuous – storage and conduction of heat can take place in every part of the wall.

Modelling of the thermal behaviour of the wall by a RC-network means a discretization of the temperature field within the wall. A certain number of so-called temperature nodes will be set up. Each node has its own time-dependent temperature progression. Between two adjacent nodes the temperature field is linear. The nodes are connected by thermal conductances. Each node has a certain thermal capacity. Storage of heat is only possible at the nodes. Therefore, the heat capacity of the wall will be discretized too.
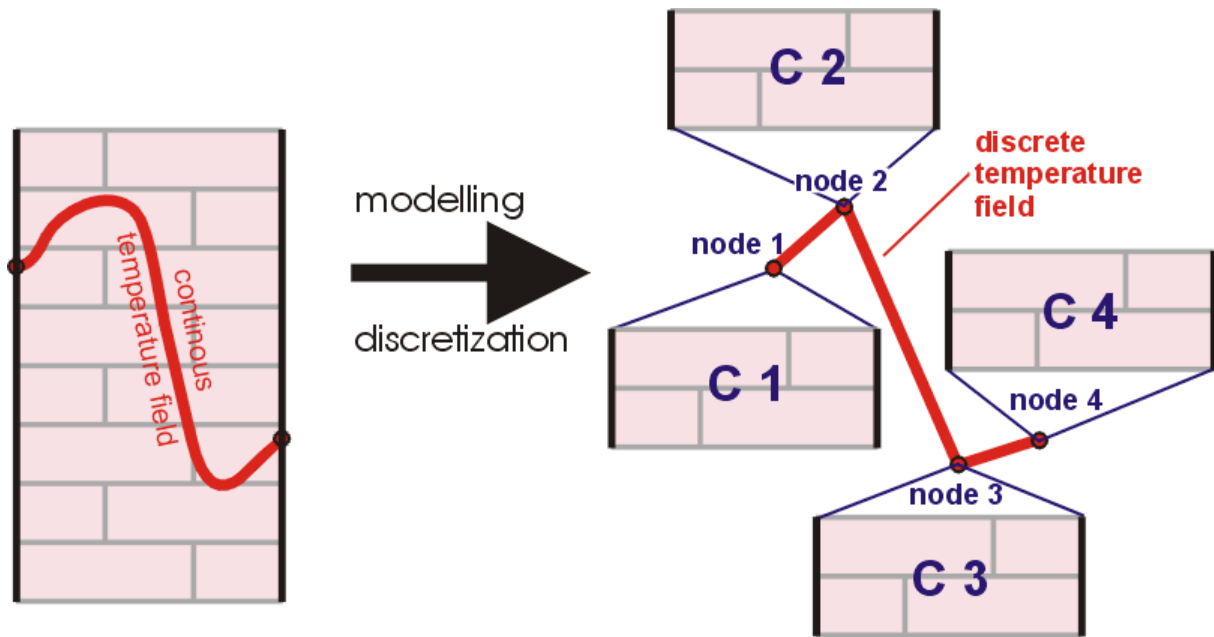
Additional heat sources (heaters, solar radiation etc) inside the thermal system or acting from outside on the system put their power in one or several nodes. Sometimes the value of their power is unknown, e.g. if solar radiation comes through a window into the room. Outside the room this solar radiation can be measured by a solarimeter (in $W/m^2$). But this heat flux density must be multiplied by the effective aperture area of the window. This aperture area, which considers also the optical losses of the glasses, is unknown in most cases and shall be determined by the experiment. Therefore, all heat fluxes entering the system always get an aperture factor A. In case of a known heating power of a radiator this factor must get a fixed value $A = 1$, whereas in case of solar radiation the value of this factor will be calculated by the program.

During the experiment the system must be thermal excited, i.e. temperature differences must be generated steady-state or dynamically. At certain positions of the system temperatures or heat fluxes must be measured. Thus, at some of the nodes of the model the temperature progression is known. The temperatures at the other nodes must be calculated by the program.

One of the nodes with measured temperature progression must be selected as the output node. The program calculates the temperature at this node and compares it with the measured one. Now the unknown values of the model components (the variable parameters) are varied as long as the best possible conformity between both temperature curves is reached.

# Examples

The successful use of the program critically depends on the choice of a suitable model. Neither the program is able to check completely if the used model is correct nor if it is qualified for the present thermal system. Here the knowledge and experience of the user are important.

Now some basics of modelling will be shown by means of some examples.

## Example 1: A simple wall

A simple homogenous wall can be modelled by a single RC-string. The number of conductances used depends on the thermal mass of the wall to be modelled. If this thermal mass is completely unknown the optimal length must be tried out. From experience, three nodes inside the wall are sufficient in most of the cases. At nodes at the margins of the model (at the borders of the system) the temperatures or the heat fluxes must be known (measured).

The following figure shows a correct model for a simple wall. The borders of the system are the nodes N 1 and N 4. At these nodes the temperatures were measured (at N 1 the outside temperature $T_{ext}$ and at N 4 the inside temperature $T_{int}$). Now the program is able to calculate the temperature progression at the nodes N 2 and N 3.
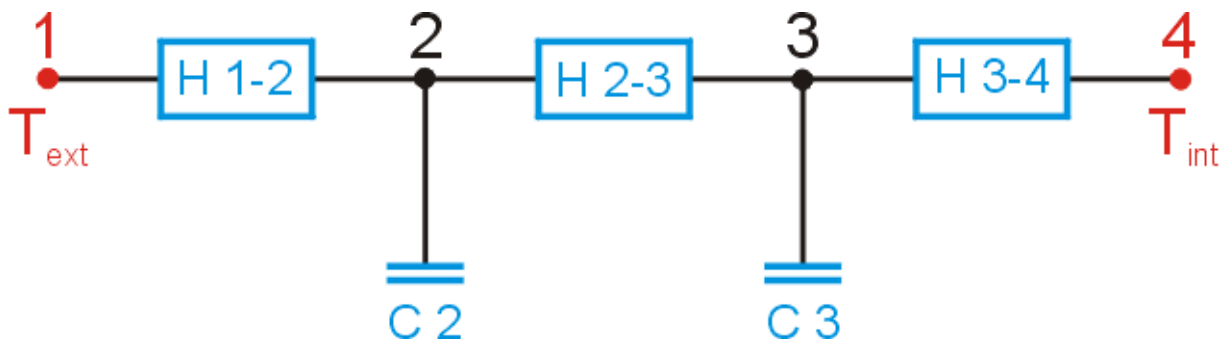


Alternatively, instead of the temperature of the node N 4 also the heat flux (positive or negative) at this node can be used, e.g. measured with a heat flux meter mounted on the interior surface of the wall.
The same applies for node N 1. But because of the difficulty to measure a heat flux on an exterior surface this version is less common.
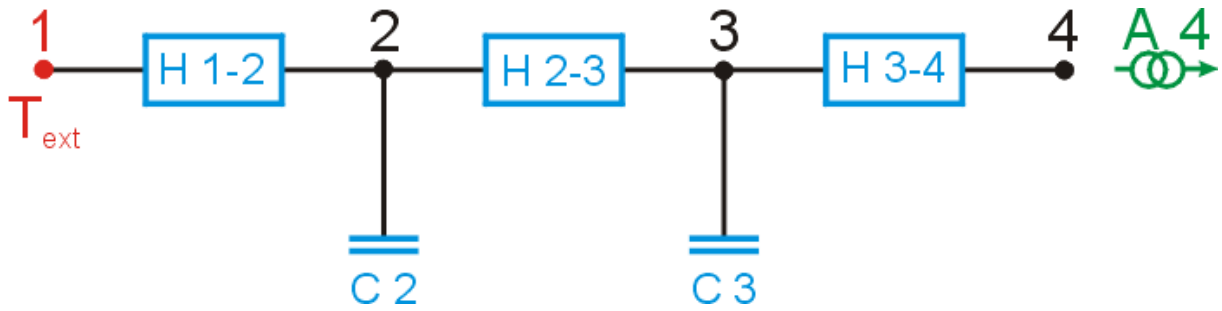
Both of the shown models could be used. In order to identify the unknown values of the components of the model (the variable parameters) an additional temperature or heat flux must be measured (output) and compared to the calculated value. In the present case the temperature or the heat flux on node N 4 could be used as output. The following two figures show the two possibilities.
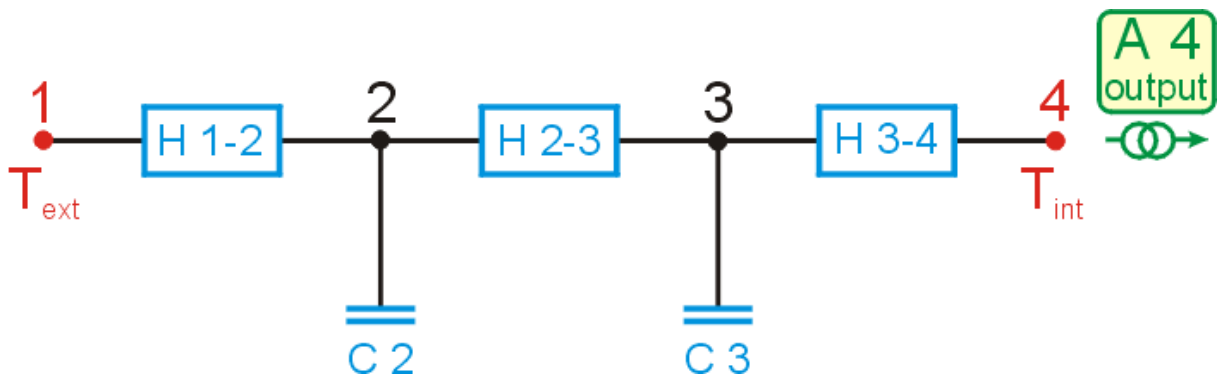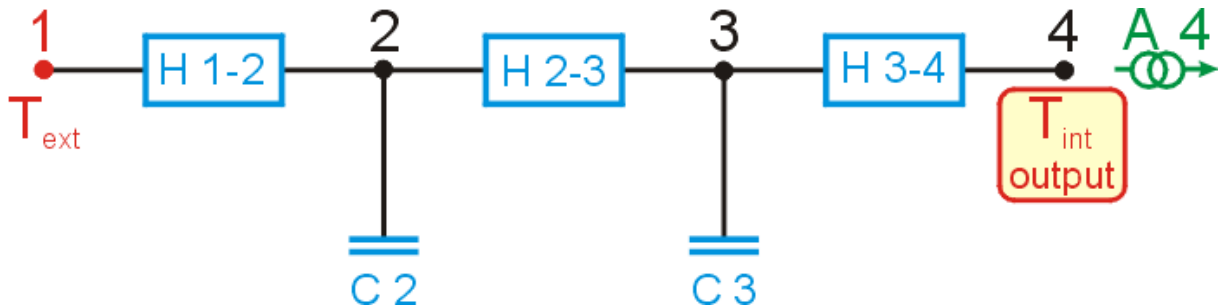




For the experimental setup both models are equivalent. Measured values are the temperatures on both sides of the wall and the heat flux of the interior surface. The model in the first figure uses the temperature $T_{int}$ as output whereas the model in the last figure uses the heat flux as output. It is advisable to try out both variants and to compare the results.

As a result of the parameter identification the values of the three conductances H 1-2, H 2-3 and H 3-4 are known. Now the thermal resistance R can be calculated:

$$R = \frac{1}{H\ 1\text{-}2} + \frac{1}{H\ 2\text{-}3} + \frac{1}{H\ 3\text{-}4}$$

Instead of the surface temperatures it is also possible to measure the air temperatures on both sides of the wall (e.g. ambient air temperature and room temperature) and to link this temperatures to the nodes N 1 and N 4. In that case the sum of the three conductances would give the UA-value:

$$\frac{1}{UA} = \frac{1}{H\,1\text{-}2} + \frac{1}{H\,2\text{-}3} + \frac{1}{H\,3\text{-}4}$$

If the exterior side of the wall is supposed to the solar radiation then two cases have to be distinguished: If node N 1 is linked to the surface temperature the solar radiation need not to be taken into account, since the temperature sensor on the surface measures the temperature increase caused by the solar radiation. Of course the sensor must have the same optical properties like the wall. But if node N 1 is linked to the ambient air temperature the solar radiation has to be considered as an additional heat source within the system. To include this into the model an additional node (N 5) between nodes 1 and 2 could be created and linked to the surface temperature. Then the solar radiation will be put into this node. Because of the unknown optical properties of the wall the aperture factor A 5 will be taken as a variable parameter.
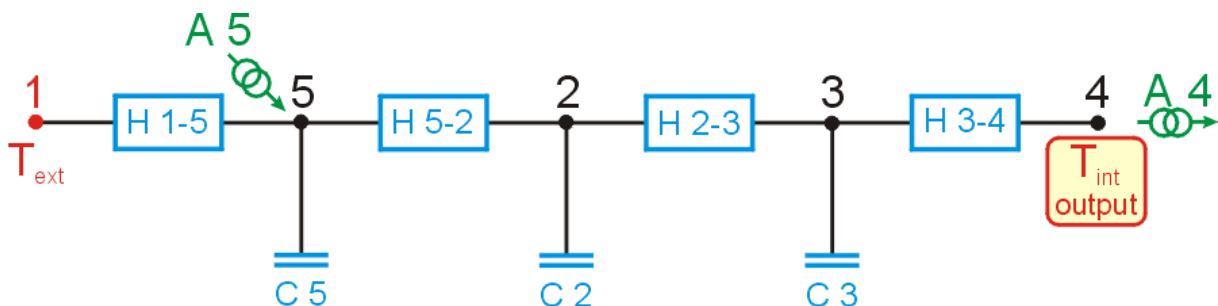


The numbering of the nodes is absolutely arbitrary.

Summary: The first step is to define which parts of the thermal system shall belong to the model and where are the borders of the model. Then it has to be checked where the system has a connection to the adjacency. On all nodes with a connection to the adjacency the temperature or the heat flux must have been measured. Remark: On nodes with known temperature capacitances have no influence.

## Example 2: Classical PASSYS test cell

A classical PASSYS test cell consists of a test room and of a service room at the northern side. The test component will be mounted on the southern side. West and east wall of the test room as well as floor and ceiling consist of the same

material and have the same temperature at their exterior side. Therefore these four walls can be modelled with one single string:
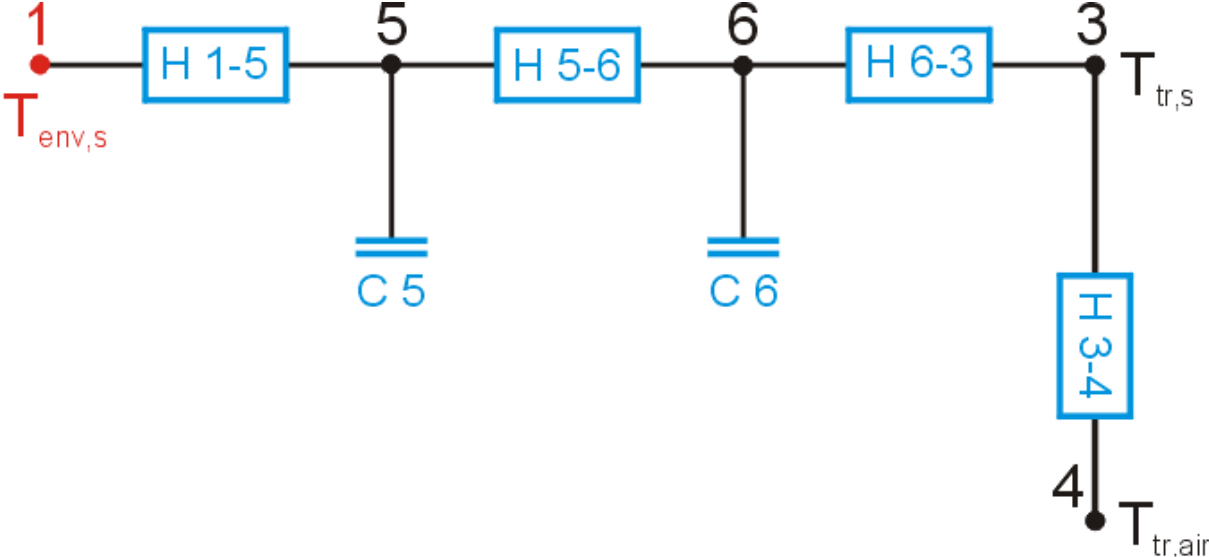


The temperature at node 1 is the average surface temperature of the four exterior walls. Node 3 is linked to the interior surface of these walls and node 4 is linked to the average air temperature within the test room.

The northern margin of the test room is modelled by a single string too. This string ends at node 2 – this is the air temperature in the service room.
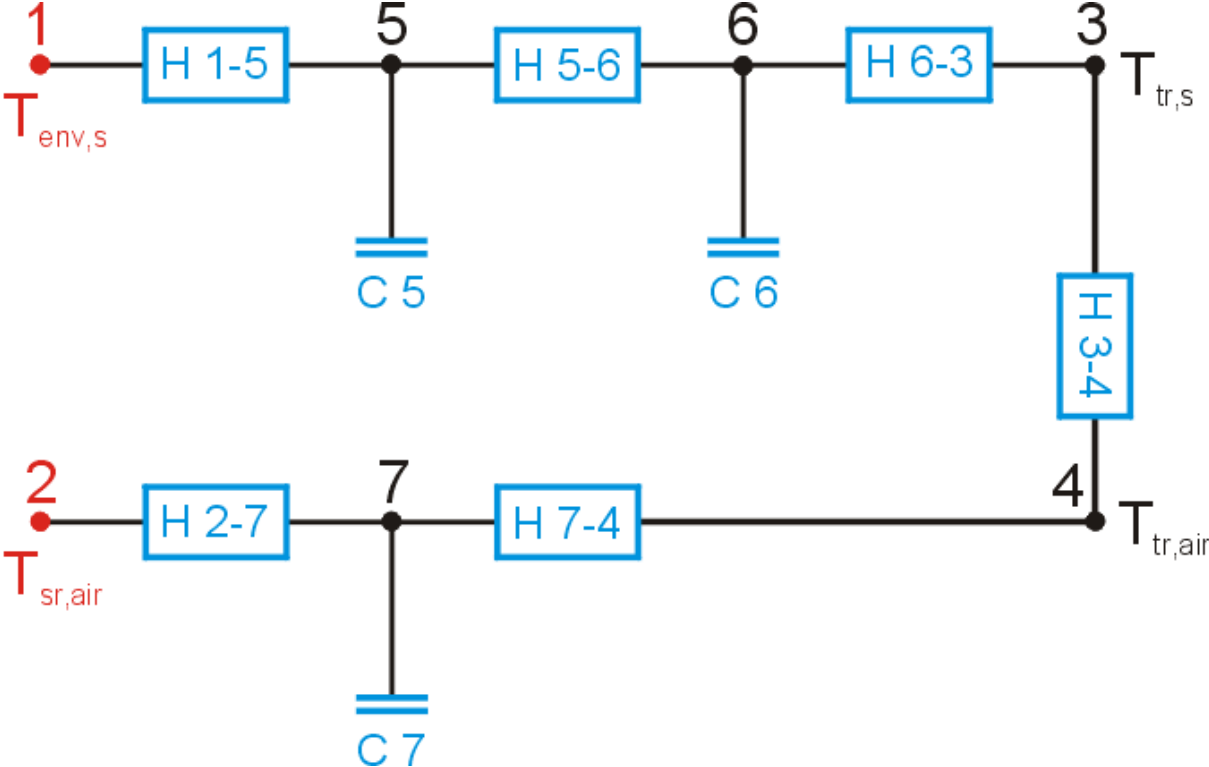


Now the test component can be connected with the air node (N 4). The model for the test component depends on the construction of this component. This shall

be exemplified by the classical reference wall. This wall is a lightweight construction with a central window.
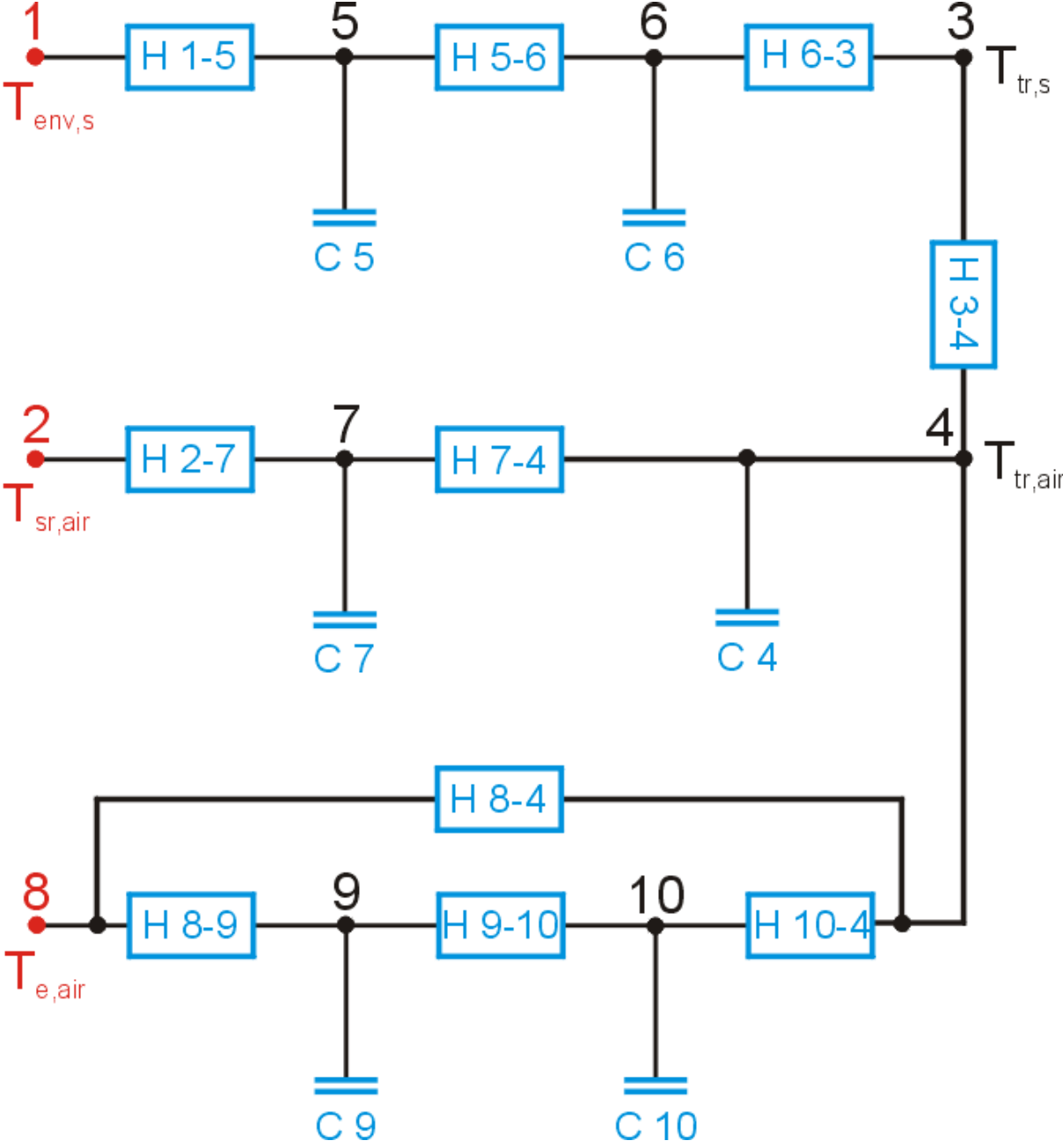


The additional parallel connected conductance H 8-4 between node 8 and 4 shall allow for thermal conduction without storage (through the window).

A parallel connection of several strings does not mean that after the identification the parts of the model can be clearly associated to certain parts of the test component. The outcome of a PASSYS test is a global result for the whole test component.

The model shown in the last figure must be completed by the measured heat fluxes and by an output value. The power of the PASSYS heating and cooling

system (Phe-Pco) is given to the air node N 4. The corresponding aperture A 4 can be fixed at A 4 = 1. The solar radiation (Rad) partly is absorbed by the wall and by the window and partly goes through the window and is absorbed within the test room. Therefore in the model the solar radiation is linked to the nodes 9 and 3. The corresponding aperture factors A 9 and A 3 are variable parameters, they will be identified during the calculations. The values of these parameters are needed for the calculation of the gA-value.

The output value usually is the air temperature of the test room (node 4). The next figure shows the complete model.
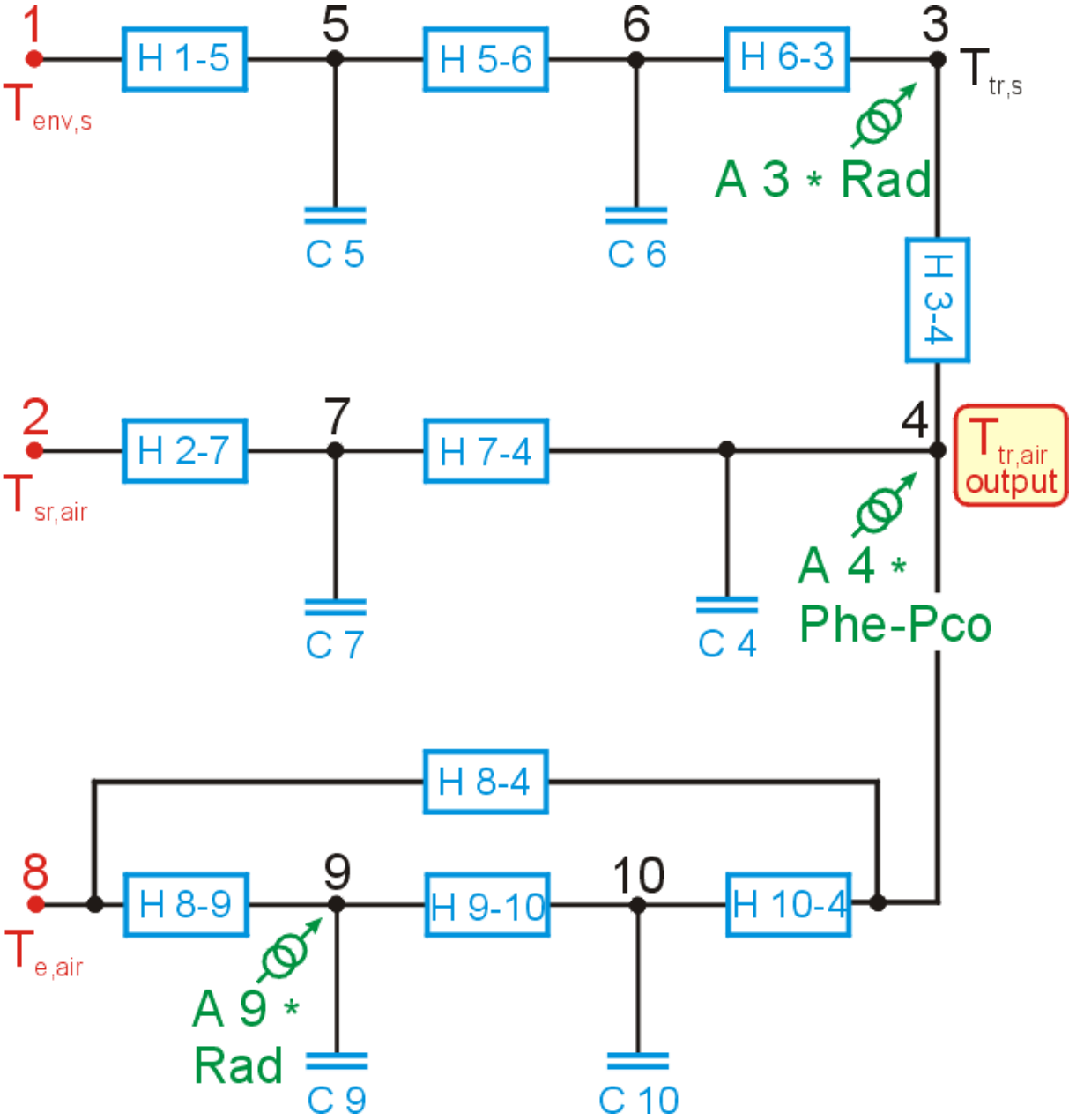


Remarks: The conductances H 1-5, H 5-6, H 6-3, H 3-4, H 2-7, H 7-4 and the capacitances C 5, C 6 and C 7 are known, they were identified during the calibration of the test cell. Node 3 corresponds to the interior surface of the test room. Though the temperature of this surface is measured, it is not used in this

example. It could be used as output value instead of the air temperature. Furthermore both temperatures (node 3 and node 4) can be used as multi-output. In LORD up to three outputs can be used together, their influence can be weighted.

## Example 3: PASLINK test cell with HFS tiles

In a PASLINK test cell with Heat Flux Sensitive tiles (HFS tiles) the total interior of the test room is clad with special heat flux meters. Heat exchange between the test room and the environment is only possible either through the test component or through these heat flux meters. The modelling of such a test cell is simple. Only two temperature nodes are needed – one for the air temperature and one for the interior surface temperature. This interior surface is one border of the system – the heat flux through this border is measured. The walls of the test cell are outside of the modelled system.
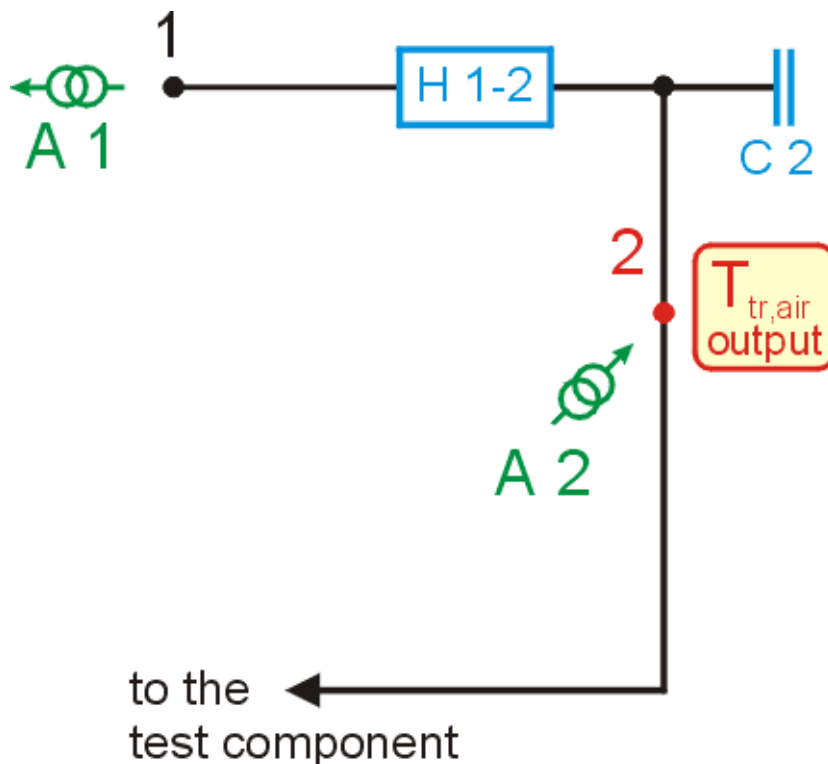


The power on node 1 is the whole heat flux measured by the HFS tiles. The power on node 2 is the heating or cooling power of the test cell. The value of the conductance H 1-2 and of the capacitance C 2 must be obtained by a calibration experiment. Also the aperture factor A 1 is identified during the calibration. Theoretical the value should be A 1 = -1 (if positive values of the heat flux are used). Practical some minor derivations are possible. Thus, with this aperture factor a certain vernier adjustment of the calibration of the HFS tiles can be obtained.

If the test component has transparent parts the solar radiation must be linked to the air node A 2. In LORD every node can only be linked with one aperture. Therefore, an additional "dummy" node N 3 must be included and connected with N 2 by a fixed "dummy" conductance  of e.g. H 3-2 = 5000 W/K. Now the heating and cooling power of the test cell can be linked to node 3 and the solar radiation can be linked to node 2.
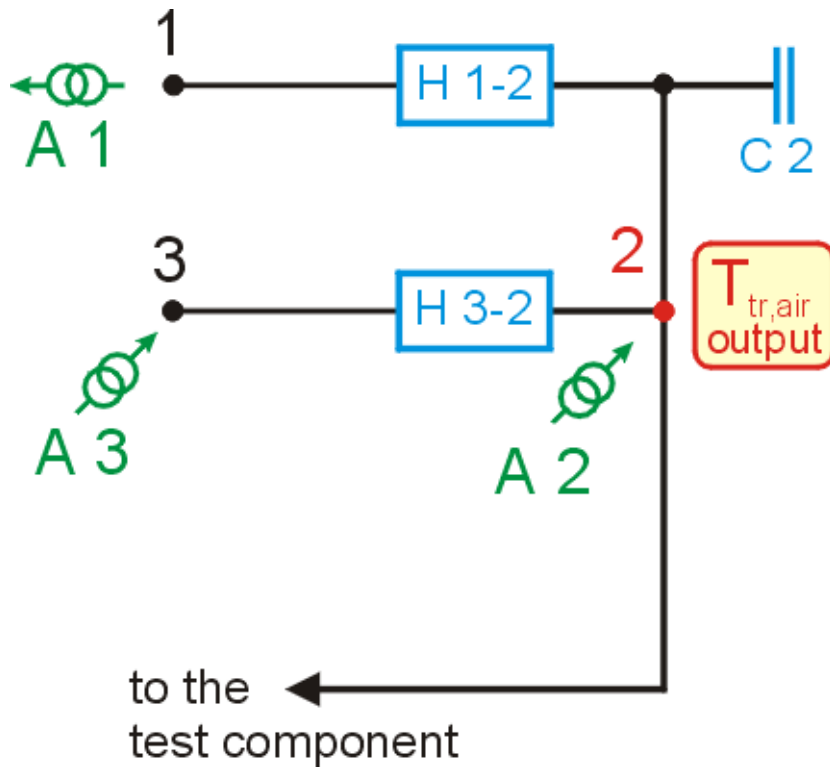


# Mathematics

For each node of the model a balance between the incoming and outgoing heat fluxes can be made. As example the following figure is used.
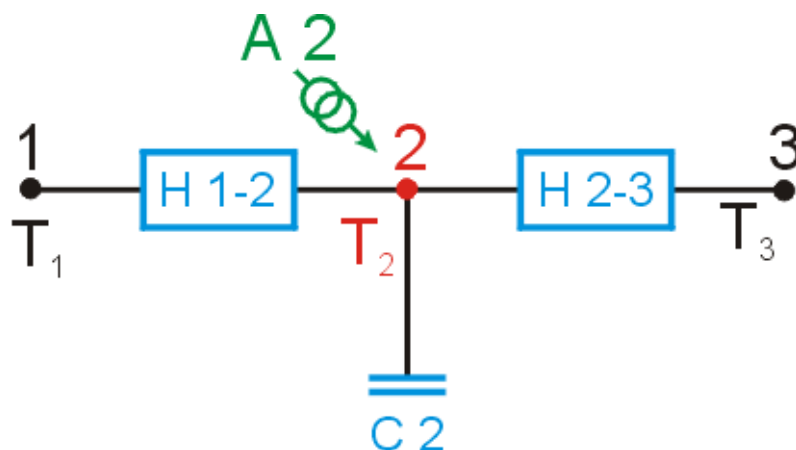
The temperature progression on node 2 shall be observed. If there is a temperature difference between node 2 and its adjacent nodes, heat (positive or negative) flows from these adjacent nodes into node 2. Additionally there is a possible external heat flux via A 2. The sum of all these heat fluxes causes a temperature change of node 2. Its strength depends on the capacity C 2. The heat balance on node 2 is:

$$\frac{dT_2}{dt} \cdot C_2 = (T_2 - T_1) \cdot H_{1\text{-}2} + (T_2 - T_3) \cdot H_{2\text{-}3} + Q_2 \cdot A_2$$

Such a differential equation is valid for every node with unknown temperature. The solution of this system of differential equations is done numerically by using the approximation

$$\frac{dT_2}{dt} \quad \rightarrow \quad \frac{\Delta T_2}{\Delta t}$$

Thus all unknown node temperatures can be calculated and the calculated output value can be compared with the measured one. As a measure for the deviation between both values the RMS (root mean square) is taken:

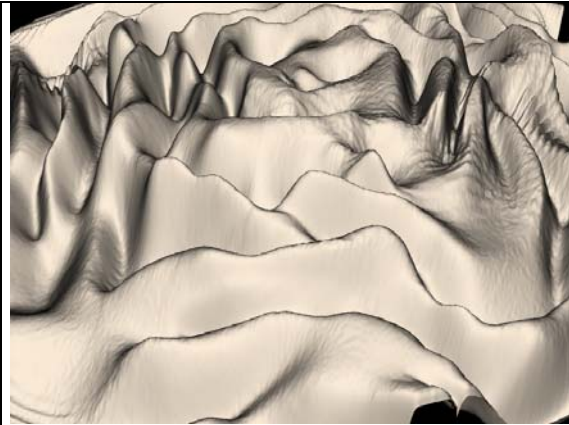$$V = \sqrt{\frac{1}{N} \cdot \sum (T_{meas} - T_{calc})^2}$$

Here N denotes the number of measured data points.

For the best possible conformation between calculated and measured output this value must be minimized. That means the parameter vector (named "Par") must be varied as long as the minimum of the function V(Par) is detected. In fact this minimization is the critical part of the calculations. Because of the unavoidable noise and of other error sources the function V(Par) has besides its global minimum a lot of local minima. The special appearance of the multi-dimensional area V(Par) strongly depends on the used model, on the number of variable parameters and on the quality of the measured data. There is no mathematical procedure which can doubtless state whether the observed minimum is a local one or the global one. A suitable choice of the starting values is very important for the success of the minimization process.

The following two examples for V(Par)-areas shall illustrate two extreme situations:
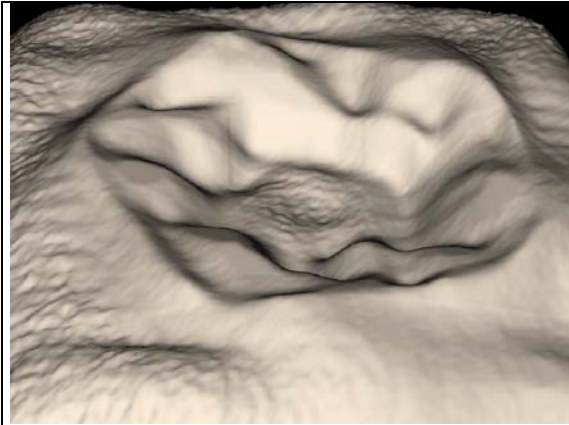
| structure of the V(Par)-area is smooth, program should find the global minimum without problems after short time | cleft V(Par)-area, e.g. caused by noised data, a lot of local minima, user should try several starting values |
| --- | --- |

In LORD two different minimization procedures are available:

## Downhill Simplex Method

This is a simple and robust procedure. In most cases it finds roughly the minimum of V(Par). The method is described in detail in "Press et.al., Numerical Recipes, Cambridge 1989".

## Monte-Carlo Method

This procedure scans the adjacency of the actual minimum (possibly a local one). The values of the actual best parameter vector are varied by means of random numbers and the respective function V is calculated. As soon as a better minimum is detected this new minimum is taken as the new starting point for the further search.

# Confidence intervals

The parameters alone do not include information about the quality of the estimation. Therefore it is important to calculate confidence intervals which contain the real values with some statistical certainty of e.g. 95 %. Such a confidence interval can be defined by Iso-V-curves, i.e. the function V for all parameter vectors within the confidence interval does not exceed a certain value $V_{Max}$. This value can be calculated according to (Téllez, in "Workshop on application of system identification in energy savings in buildings", pp. 281, edited by J.J. Bloem, 1994):

$$V_{Max} = V_0 \cdot \sqrt{1 + F(m,N,s) \cdot \frac{m}{N-m}}$$

$V_0$ ... minimum of V(Par)
m ... number of variable parameters
N ... number of data points
s ... degree of the statistical certainty
F ... Fischer-distribution (can be obtained from tables)

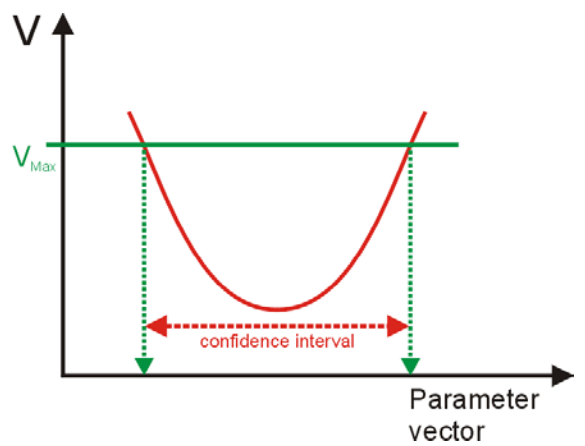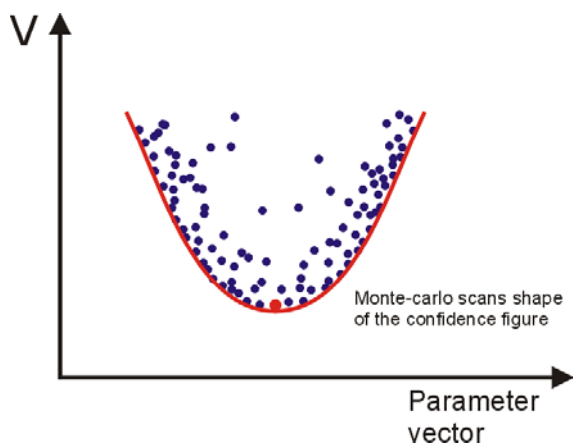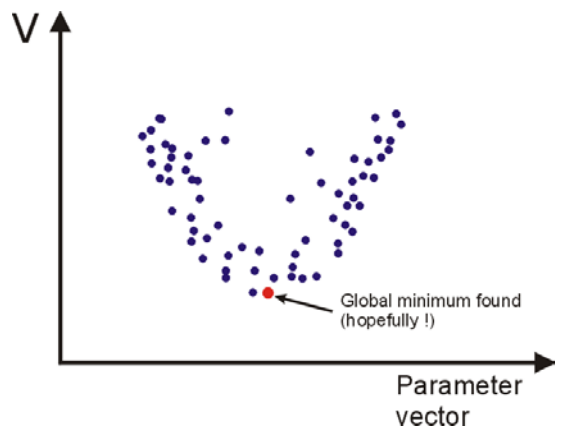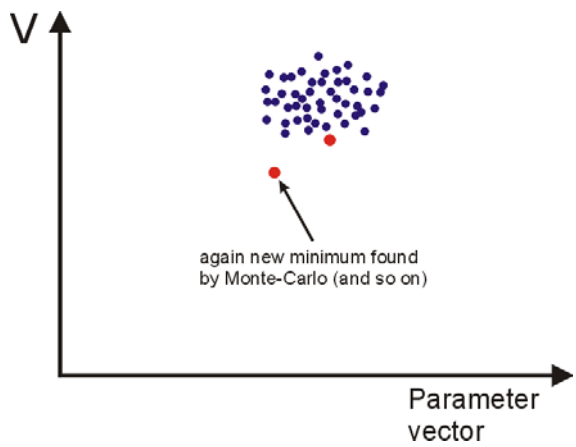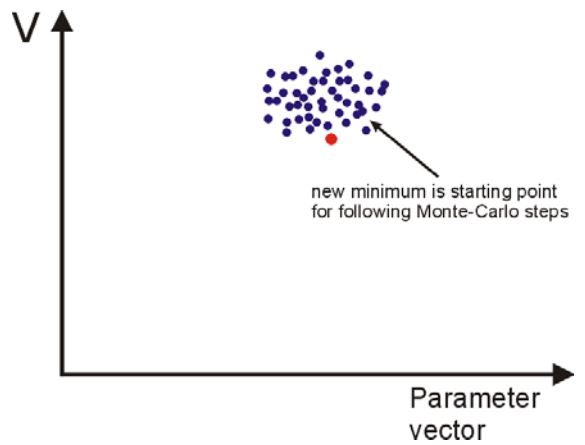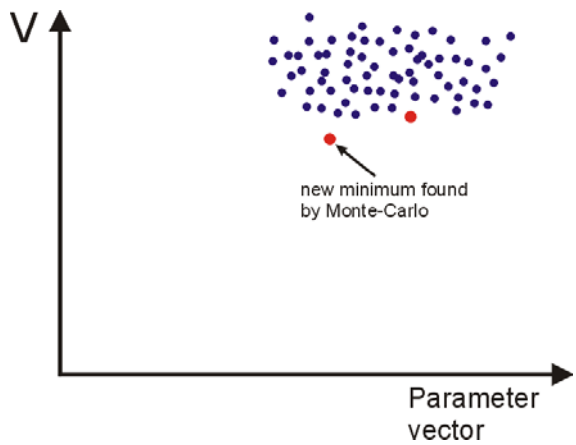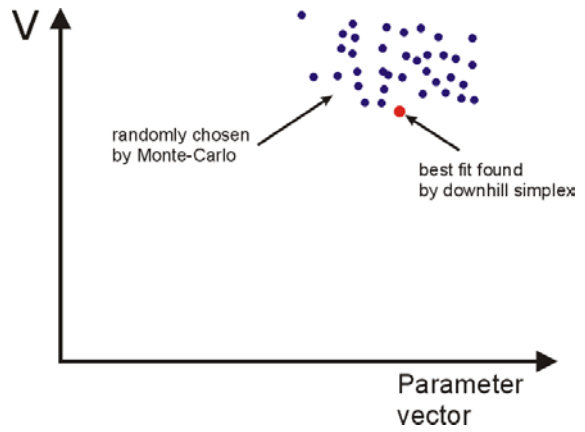From this equation it could be deduced that the confidence intervals can be lowered to any degree by an increase in the number of data points, independently whether this increase is realized by an increase in the total duration of the measurement or by a decrease of the measuring interval. However, because of the inertia of every thermal system the additional information get by a decrease of the measuring interval is limited. In case of too short measuring intervals the data are autocorrelated. This autocorrelation has to be taken into account. It can be done by replacing the real number N of data points by an effective number $N_{eff}$ (van der Linden and van Dijk: "MRQT User Guide", TNO Delft, 1993):

$$N_{eff} = N \cdot (1 - |A|) + 1$$

A is the autocorrelation in the residuals $\Delta R$ (deviation between calculated and measured output value) over one time step:

$$A = \frac{\sum_{j=1}^{N-1}(\Delta R_j - \overline{\Delta R}) \cdot (\Delta R_{j+1} - \overline{\Delta R})}{\sum_{j=1}^{N}(\Delta R_j - \overline{\Delta R})^2}$$

The confidence intervals of the individual parameters and of the UA- and gA-value were calculated during the Monte-Carlo procedure. The Monte-Carlo procedure scans the adjacency of the minimum – therefore it is able to detect all parameter vectors with V(Par) < $V_{Max}$ as well.

best fit found
by downhill simplex

randomly chosen
by Monte-Carlo

best fit found
by downhill simplex

new minimum found
by Monte-Carlo

new minimum is starting point
for following Monte-Carlo steps

again new minimum found
by Monte-Carlo (and so on)

Global minimum found
(hopefully !)

Monte-carlo scans shape
of the confidence figure

$V_{Max}$

confidence interval
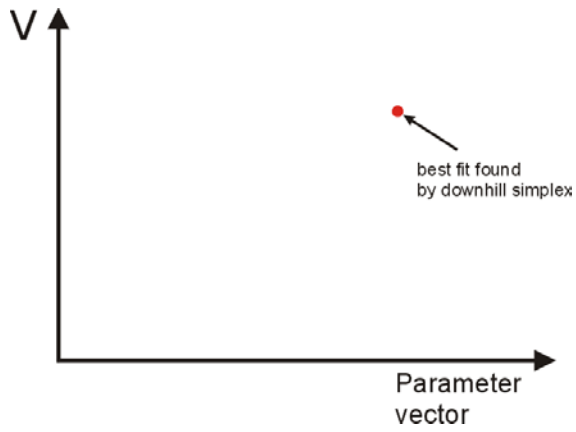
The projection of the function V(Par) at the position V = V$_{Max}$ onto the parameter space (this space has as much dimensions as the number of variable parameters) defines the confidence interval.

The following example shows this projection for the case of only two variable parameters:
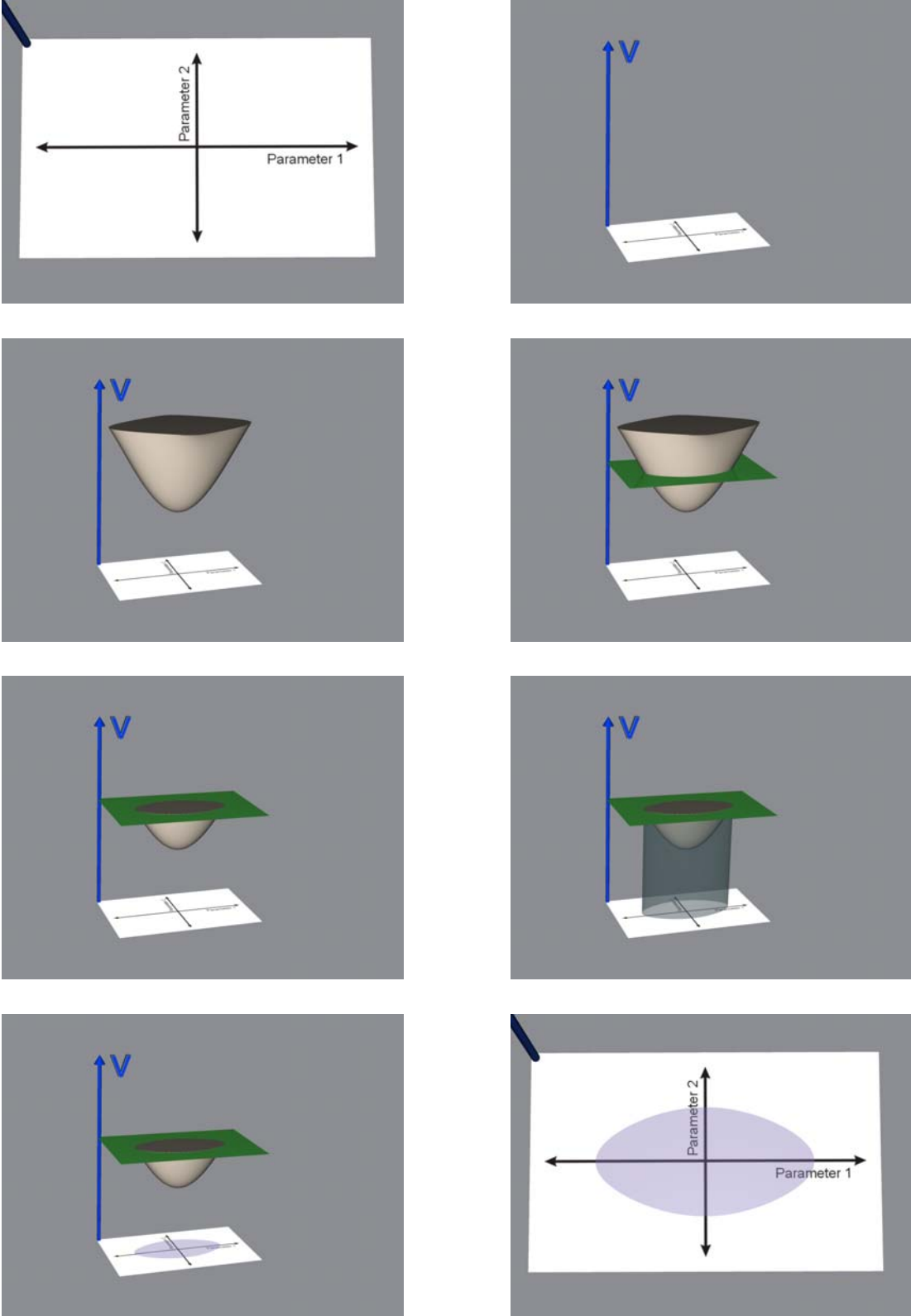
The confidence figure of these two parameters is an ellipse. Their main axes are parallel to the axes of the coordinate system. The true parameters are within this ellipse with an probability of 95 %.
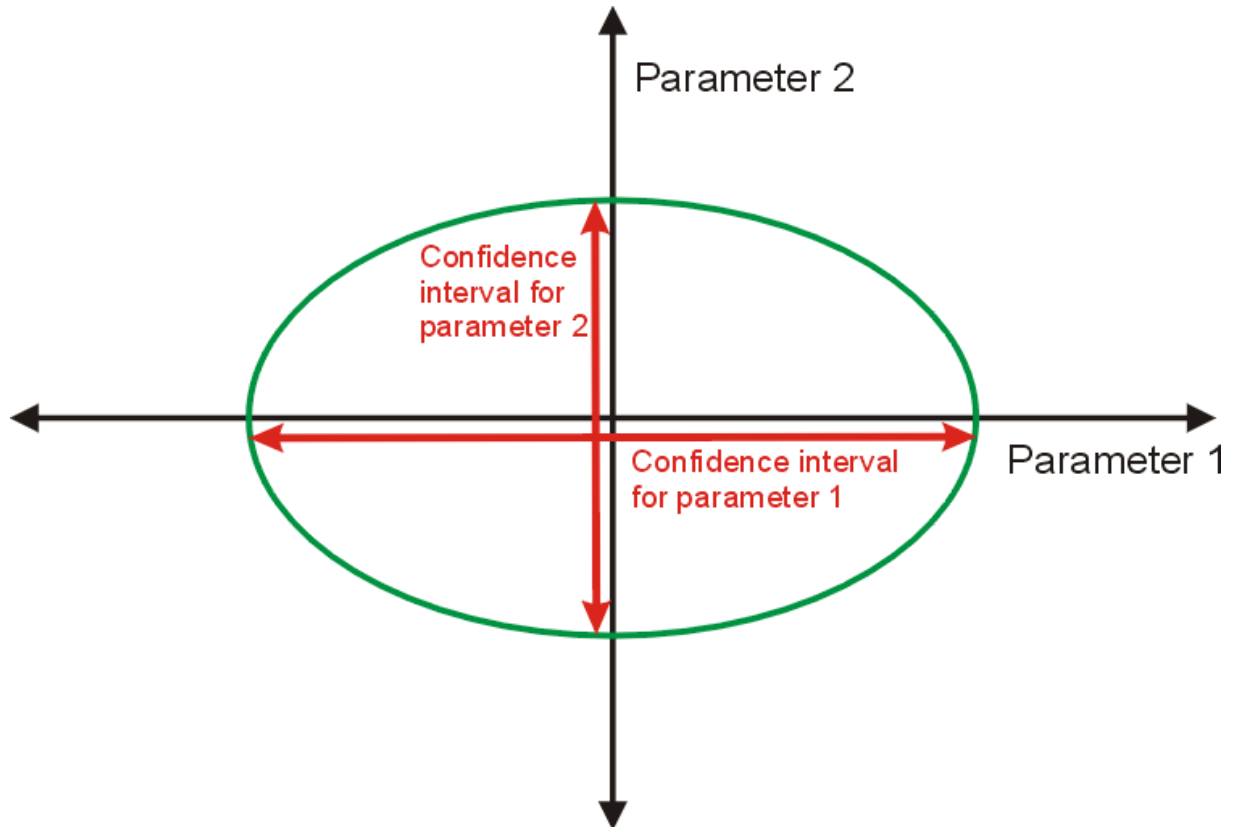


# Correlation of parameters, statistics

Sometimes the confidence intervals of individual parameters can become very large. However, this does not mean that the quality of the fit is bad or that the final results (UA-value, gA-value) have a large error. The reason can be a correlation between individual parameters. This shall be demonstrated by a simple example:

A homogenous and massless wall shall be considered. Such a wall could be modelled by only one single conductance:
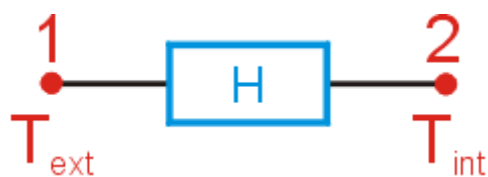
If the temperatures $T_{ext}$ and $T_{int}$ are the air temperatures on both sides of the wall then the conductance H corresponds to the UA-value of the wall: UA = H. Also the confidence interval of H equals to the confidence interval of UA.
Of course it is also possible to model the same wall by a series connection of two conductances $H_1$ and $H_2$:
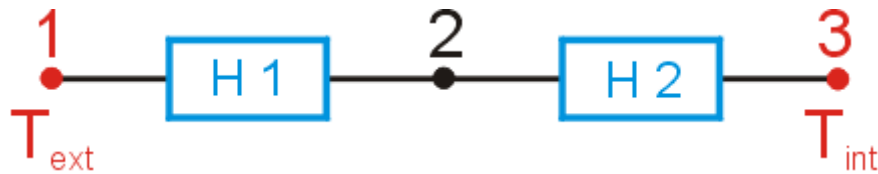


Surely the result UA = $H_1$ + $H_2$ should be the same as in the previous case. But there is a big number of combinations $H_1$/$H_2$ which all give the same result. Therefore, the confidence intervals of these parameters will be very large. Nevertheless the confidence interval of UA should be not larger than in the other case. The explanation for this behaviour is that the parameters $H_1$ and $H_2$ are correlated, i.e. their values are not independent. If one of the conductances has a high value than the value of the other conductance must be small and vice versa. This correlation can also be seen in the confidence figure of theses two parameters. It is still an ellipse, but their main axes are no longer parallel to the coordinate axes:
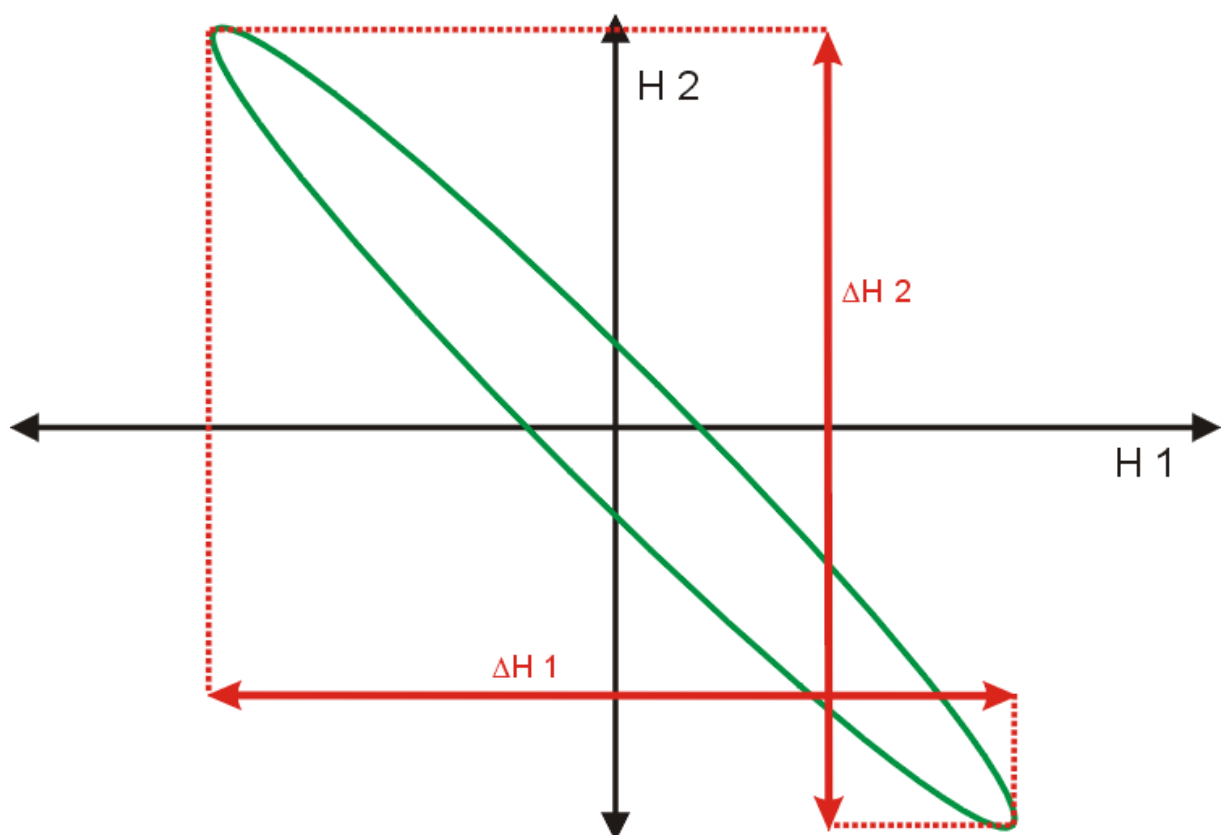


A measure for the correlation between parameters is the correlation coefficient. It is zero if the parameters are uncorrelated. If the parameters are completely

correlated it's value is ± 1. In this case changes in one parameter can be completely compensated by a respective change in the other one.

A model of a thermal system would be perfect if there were no correlation between the parameters. Of course in practice this is unattainable. Mostly the correlations are not as obviously as in the example. For this reason LORD calculates some statistical information what can help to optimize the model. The correlation coefficients between the parameters are shown in a table. The calculation only proceeds during the Monte-Carlo simulation. This procedure permanently creates new parameter vectors Par and calculates V(Par). If the condition $V(Par) < V_{Max}$ is fulfilled the respective parameter vector is stored in the so-called observation matrix X. This matrix has as many columns as the number of variable parameters and as many rows (n) as the number of parameter vectors with $V(Par) < V_{Max}$. The variation matrix M is obtained by subtracting the mean values of each column from all elements of the matrix X. The matrix M has the same dimension as the matrix X. Now the matrix product

$$T = M^T \cdot M$$

can be calculated. This new matrix T is symmetrical. If all elements of T are divided by the number of parameter sets then the covariance matrix S is obtained. Their diagonal elements are called variances, the other elements are called covariances. Finally, the correlation matrix R can be calculated by:

$$R_{jk} = \frac{t_{jk}}{\sqrt{t_{jj}} \cdot \sqrt{t_{kk}}}$$

The dimension of this symmetric matrix is equal to the number of variable parameters. The diagonal elements must be equal to 1. The other elements correspond to the correlation coefficients between the individual parameters. In LORD the values of the matrix R are displayed permanently on the "Statistics" page (selection "cross correlation"). The more parameter sets with $V(Par) < V_{Max}$ are founded the more reliable are these values.
As soon as a new minimum of V(Par) is detected all matrices were deleted and then recalculated.


# Principal component analysis

The model of the thermal system should be as simple as possible and should contain the smallest possible number of variable parameters. The more variable parameters have to be calculated the more time the calculations will take and the

more incertain are the results. If there is a significant correlation between parameters then the model has more variable parameters than needed. Furthermore there could be parameters which have only a minor effect on the final result. The identification of such parameters also consumes computer power.

The principal component analysis (PCA) gives an informative overview about the influence of the parameters and their correlations among each other. It's basics shall be explained again by means of the last example (series connection of two conductances).

As already mentioned a model without parameter correlations would be perfect. The main axes of the confidence ellipse then should be parallel to the coordinate axes. In the last figure this is not the case. To get more information about the properties of the parameters now a transformation of the coordinate system is undertaken. The existing coordinate system is rotated as long as it's axes are parallel to the main axes of the confidence ellipse. In the example there are only two parameters and hence the coordinate system is two-dimensional. Therefore the transformation is a simple plane rotation:
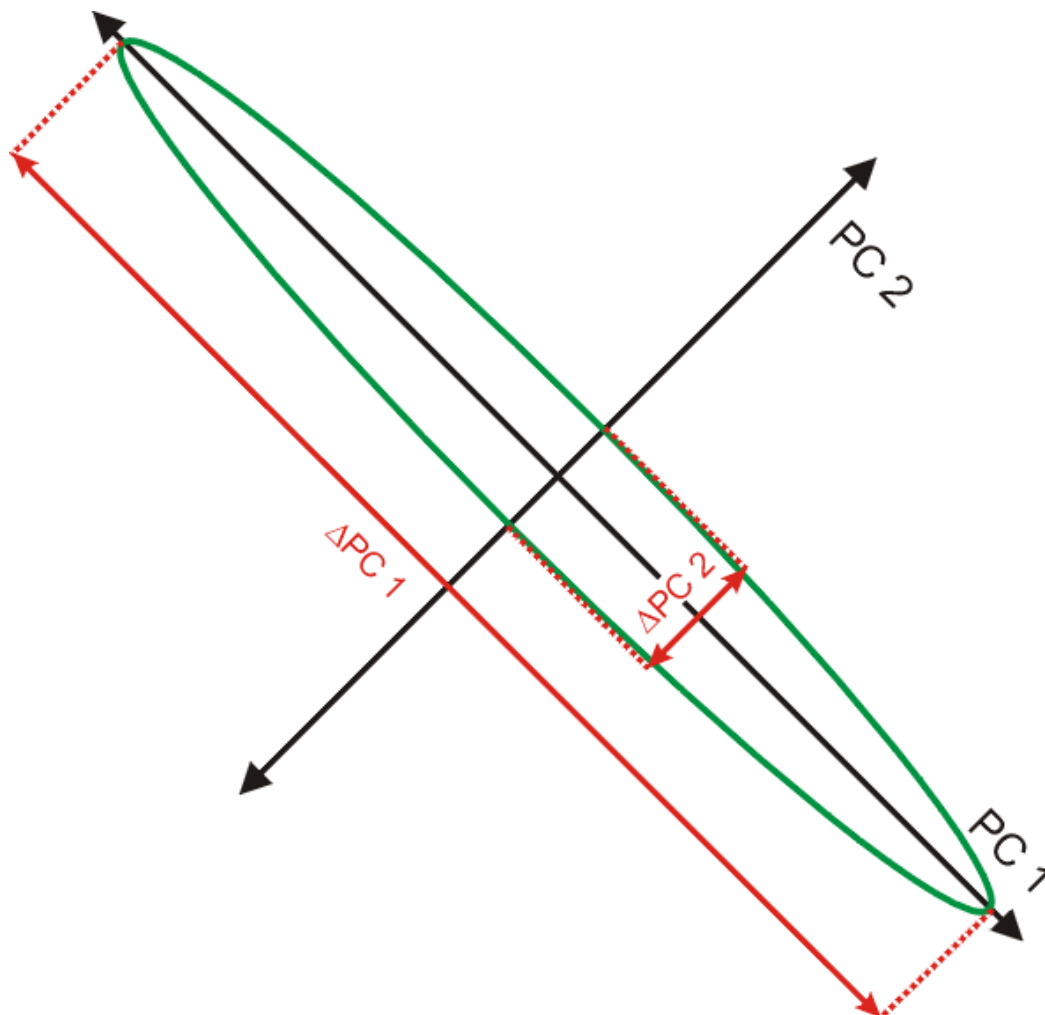
In the new (transformed) coordinate system all parameters are uncorrelated. However, the axes of the new coordinate system are no longer the original parameters, but linear combinations of these parameters. These linear combinations are called principal components of the correlation matrix. Mathematical the principal components are the eigenvectors of the correlation matrix. The corresponding eigenvalues specify the influence of the particular principal component on the result.

In LORD the principal components were displayed together with their eigenvalues on the "Statistics" page, optionally in a table or in a chart. The principal components are ordered by their eigenvalue, i.e. by the strength of their influence.

|  |  | Param. 1 | Param. 2 | Param. 3 | Param. 4 | Param. 5 |
|---|---|---|---|---|---|---|
|  | Influence | H 02-01 | H 03-02 | H 04-03 | C 02 | C 03 |
| Number 1 | 1.5608 | 0.7054 | -0.7044 | -0.0518 | -0.0369 | 0.0467 |
| Number 2 | 1.2983 | -0.0245 | -0.0307 | 0.7100 | 0.0105 | 0.7030 |
| Number 3 | 1.1714 | 0.0378 | -0.0578 | 0.3545 | 0.8551 | -0.3720 |
| Number 4 | 0.5283 | 0.0364 | -0.0211 | 0.6057 | -0.5167 | -0.6037 |
| Number 5 | 0.4412 | 0.7064 | 0.7065 | 0.0262 | 0.0182 | 0.0287 |

Example for the table of principal components: Each component is a linear combination of the original parameters, the coefficients of these combinations are displayed in the table.
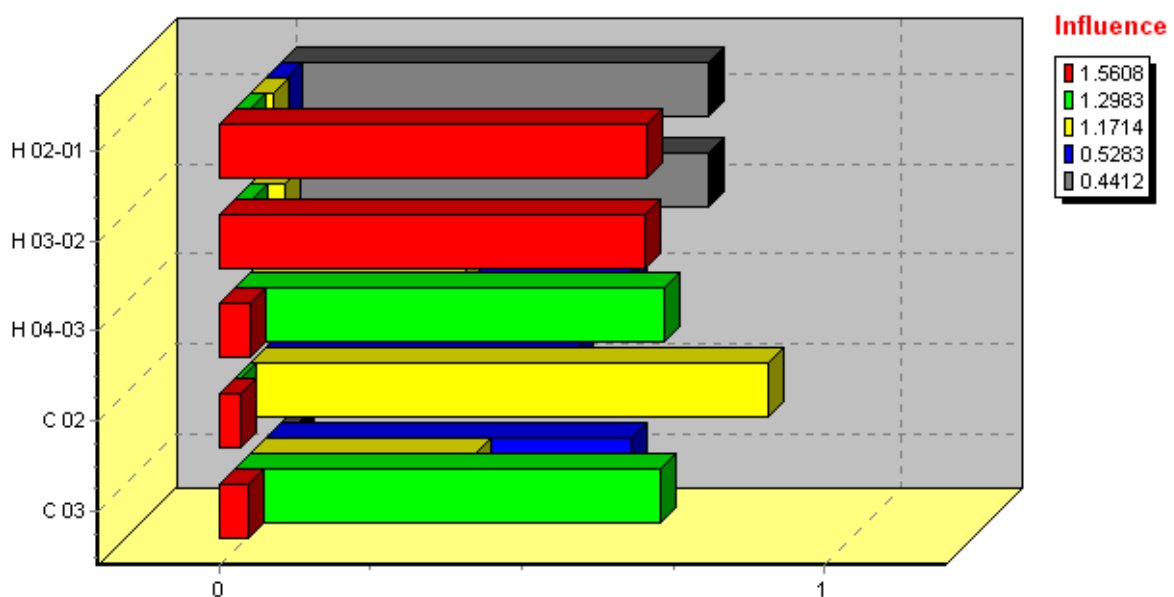


The same example as a chart – here the absolute values of the coefficients are displayed

The interpretation of the principal component analysis needs some experience. Firstly the simple example (series connection of two conductances) shall be regarded again. Since there are two variable parameters there are also two principle components. They shall be indicated by $PC_1$ and $PC_2$. A possible result could be:

$$PC_1 \;=\; H_1 - H_2$$

$$PC_2 \;=\; H_1 + H_2$$

The second component $PC_2$ has a practical meaning – because it is the sum of $H_1$ and $H_2$ this component corresponds to the UA-value. As can be seen in the figure with the transformed coordinate system the variance of this component is relatively small – hence the UA-value can be determined with a much higher accuracy than the individual parameters. The other principal component with a large variance corresponds to the difference of the two parameters and has no practical meaning.

In most of the cases this interpretation is not as simple as in the example. But with some experience a lot of indications for the optimization of the model can be read off:

- parameters have a significant influence on the result if they have large coefficients in the significant principal components

- if the largest coefficients of one principal component are shared over two ore more parameters then only a certain combination of these parameters has influence on this component. If this component has a large eigenvalue then it means that the number of free parameters could be reduced

- if one parameter has large coefficients only in principal components with small eigenvalues then this parameter has only a small influence on the result, it could be fixed during the identification

# Short user guide

The operation of the program will be demonstrated by an example. This example was a task in the Second System Identification Competition. In this task data from measurements of temperatures and heat fluxes at a homogenous wall were given. The name of the data file is "\Lord\Data\Data_B01.Dat". Data

files used in LORD must be pure ASCII files with data columns separated by blanks. Alternatively, CSV-files can be used.
The files must have a headline with column names in the format

"Name1"   "Name2"  ...

Duplicate names are not permitted.

The data file may contain any number of comment lines. The first line which starts with an ″ will be interpreted as the headline with the column names.
It is possible to specify the used time step in the data file. If one of the comment lines starts with the keyword 'Time' the following number will be taken as the time step (in minutes). Otherwise the default value of 60 minutes is assumed.
The time step can also be set in the "Model" page.
The first lines of the example file are:

Data from the Second System Identification Competition, Case B
Time 60
"TExtern"   "TIntern"   "Heat Flux"
3.8200000e+000  1.9090000e+001 5.8010000e+000
4.8100000e+000  1.9720000e+001 6.0670000e+000
4.8900000e+000  1.9400000e+001 5.8820000e+000
4.6400000e+000  1.9320000e+001 6.0870000e+000
....................

After starting LORD the "Data files" page will be opened. On the left side there is a panel with the name "Data file". Here existing data files can be opened. The names of their columns are displayed. On the right side there is a panel with the name "The project". At first the actual project still does not contain data. By clicking on the respective buttons under the blue arrow one or several data columns from an opened data file can be put into the project. Thus the project can contain data columns from several data files. New project columns can also be created by arithmetical operations between project columns (button "Create new columns"). This can be needful e.g. if an electrical power must be calculated from a voltage and a current measurement.

It is possible to save all the project columns in a new data file for later use (button "Save project columns").

Of course the project can contain more data columns than needed for the actual model. All data in the project are stored in the working memory of the computer, each number (each measured value) needs 8 byte.

- Click on "Open a data file"
- Select "\Lord\Data\data_b01.dat"
- Click on "All columns (... put into the project)"

Now all three data columns are in the project and are available for the calculations.

The next step is to define the thermal model. Thereto the "Model" page must be opened. As example the following model shall be created:
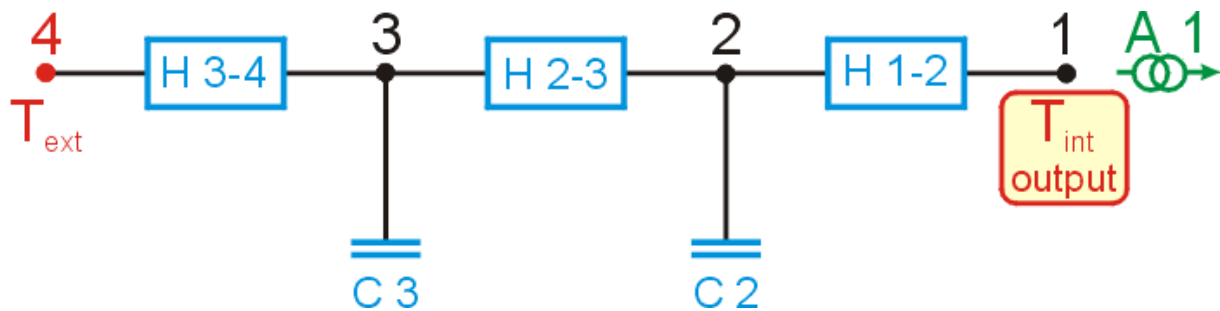


At first the required conductances will be defined. In the two blue stripes (top left) two different node numbers must be marked. The order is arbitrary – node 1 on the left stripe and node 2 on the right stripe or vice versa has the same effect. There is no need that the first used node must be node 1, e.g. it is also possible to connect node 7 with node 19. As soon as two different node numbers are marked on the two stripes and if these two marked nodes are still not connected the button "Add conductance" becomes active. By clicking on this button two new panels appear on the screen. The left one has the name "Conductances". Here all defined conductances are listed together with their starting value (default: H = 10 W/K). In this panel the starting values and the status (variable / fixed) of the conductances can be changed. The conductances can also be removed here. The second new panel is named "Capacitances". Each node what belongs to the model (i.e. each node what is connected with another) gets automatically a capacitance with the default starting value of C = 0.5 MJ/K. Capacitances can not be removed, but their starting value can be set to C = 0.

- Click on number 1 in the left blue stripe
- Click on number 2 in the right blue stripe
- Click on "Add conductance"
- The same with H 2-3 and H 3-4

Next it must be defined on which nodes temperatures and/or heat fluxes were measured. This can be done on the "Link data to nodes" page. On the left side of this page the measured temperatures were linked and on the right side the measured heat fluxes. The node which shall be linked to a measured data column must be marked on the purple stripe in the middle of the page. Then one of the buttons "Link to temperature" or "Link to flux" must be clicked. A selection window will be opened and one of the data columns (from the actual project) can be selected. If the marked node is still not connected with another (i.e. it still not belongs to the model) an error message will be displayed.

- Click on number 1 in the purple stripe
- Click on "Link to temperature"
- Select "TIntern"
- Click on number 4
- Click on "Link to temperature"
- Select "TExtern"
- Click on number 1
- Click on "Link to flux"
- Select "Heat flux"

Now the "Model" page must be opened again. Meanwhile a new panel with the name "Apertures" is displayed. Every selected heat flux gets automatically an aperture factor and all these factors are listed in this panel. Starting value (default: A = 1) and status (fixed / variable) can be changed. In the example the aperture factor of the heat flux meter is constant and known (A = 1). Therefore only the status must be set to "constant".

- Click to mark the line "A1 = 1.000, variable"
- Click on the button "fixed <--> variable"

Now the output value must be defined. Up to three different outputs can be used, even though in most cases one output is sufficient. Outputs can only be defined if nodes are linked with data columns.

- Click on the line "No.1   Not specified"
- Select "Temperature "TIntern" at node 1"

The capacitance on node 4 was set to C4 = 0.5 MJ/K and to "variable" by default. This node now is linked to "TExtern", that means that the temperature progression of this node is known. Therefore the capacitance on this node does no longer take an effect. During the parameter identification this capacitance will not be identified, even though it's status is "variable".

The temperature at node 1 was measured too ("TIntern"). But because this temperature was selected as output the temperature of this node will be calculated anyhow. Therefore the capacitance C1 has influence on the result. The choice of the starting value and of the status of the capacitance at an output node can be very sensitive. In most cases it is unfavorable to use a fixed value.

The time step of 60 minutes was already read from the data file. If could be changed by clicking on the respective number.

Now all necessary specifications are done. It is recommended to save the model (button "Save model" on the "Model" page). All information about the model will be saved, expect the time step, because it rather belongs to the data than to the model. If an existing model is loaded all previous specifications to the model will be overwritten !

Often not only the values of the individual parameters are of interest but also combinations of certain parameters, e.g. the UA- and the gA-value of a wall construction. The program can calculate this values automatically. Thereto on the "UA and gA" page it must be defined which node is the "interior one" and which is the "exterior one". If gA shall be calculated additionally the data column with the solar radiation must be specified.

- Click on "not specified" in the line "Interior node"
- Select "Node number 1"
- Click on "not specified" in the line "Exterior node"
- Select "Node number 4"

Now the identification can be started by clicking the "Start" button on the right side. The "Output graphics" page is opened automatically. Here the fitting process can be observed visually.

If more than one output has been defined it is possible to switch the individual outputs and the weighted mean output simply by clicking on the numbers right next to the graphics, separately for both the residual and the output graphics.

It is also possible to change the range and number of the fitting window(s) on this page. The identification run tries to minimize the sum of the residuals of all data points within the fitting window(s). The fitting windows are indicated by the yellow rectangles on the graphs when the "Display FW (Fitting window)" option is enabled. As many fitting windows as required can be created by the user. The following operations are possible :

- click with the right mouse button within a fitting window: number of the window and values of the left and right margin are displayed, these values can be changed numerically
- click with the right mouse button on a margin of a fitting window and drag the mouse: shifts the margin of the fitting window
- press the Ctrl(Strg)-button and click with the right mouse button within a fitting window: fitting window will be deleted
- press the Ctrl(Strg)-button and click with the right mouse button outside the fitting windows: a new fitting window will be created

All these operations can be carried out during the identification process runs. It is interesting to see the effects "online". In case of difficulties to target the margins of the windows the zoom function should be used (press left mouse button and drag the mouse, drag from left to right zooms in, drag from right to left zooms out).

In general, and depending on the system being modelled, the first hours or days of a measurements should not be within the fitting windows(s) because of the unknown initial conditions.

By clicking on one of the "Copy graph to clipboard" buttons the respective graphic will be copied as an bitmap to the clipboard. From there it can be inserted into other documents (e.g. for printing).

On the "Results" page the numerical results are displayed.
Two identification methods are implemented in the program - the Downhill Simplex method and the Monte-Carlo method. Every start or continuation of an identification process begins with the Downhill Simplex method. After a certain criteria is fulfilled the program changes automatically to the Monte-Carlo method. Nevertheless, it is possible to switch between the two methods by clicking on the arrow.

The identification process will continue indefinitely until the Stop button is pressed. It is up to the experience of the user to decide if the fit is "good enough".

If the Stop button is pressed the identification process is terminated and the best parameter values up to this time will be saved. If the identification process is restarted, these parameters will be automatically adopted as the new initial values. If the program is paused the parameter values, as well as their status (fixed or variable), can be changed.

Two output files are created in the subfolder \results\ whenever the Stop button is pressed. These files have the extensions *.Res and *.Log and the same name

as the data file currently in use, followed by the "-number", where "-number" is the lowest available number. Both files are ASCII files. The *.Res file contains a time column and the output variables (calculated and measured). This file can be used for the analysis of the results.
The *.Log file contains all the information about the model used, the initial values and the results.
It is recommended that these files are deleted or compressed occasionally.

It is possible that the identification is unsuccessfully because one ore more parameters "run away", i.e. the get unrealistic high values. Therefore on the "Limits" page their possible range of values can be limited.

The default limits for conductances are 0.1 W/K .. 2000 W/K, for capacitances 0.00001 MJ/K .. 3 MJ/K, for apertures 0.1 .. 10 and for coefficients 0 .. +1E30. The limits can be set to other values by the user. The specified limits are only active if the respective box is checked.

It is possible to both change the limits and to enable/disable them during the identification process.
When the result becomes stable and satisfying it should be tried to disable the limits.

On the "Parameter functions" page the variable parameters can be assigned to some predefined functions. Thus parameters can depend on other parameters, on temperatures on certain nodes or on values in certain data columns. The possibilities of these parameter functions are very manifold, e.g. angle dependent gA-values or temperature dependent thermal conductivities could be determined.

Every predefined function contains three new parameter coefficients: P0, PA and PB. During the identification run the value of the original parameter is calculated according to the selected function. P0, PA and PB can be fixed or variable. The status of the coefficients and their initial values can be changed by clicking on the respective places in the "Coefficients" window.
If changes are made in the model after the parameter functions have been defined, these changes could influence the definitions of the functions. It is recommended that the parameter functions are defined as the last step before starting the identification process.
To demonstrate the operation of the parameter functions it is assumed that in the previous example the values of the two conductances H 1-2 and H 3-4 shall be equal. Then the last steps before starting the identification would be:

- Mark the line "P3: H 04-03" in the right window and the function "Type 7" in the left window (parameter H 3-4 shall depend on another parameter)

- Click on the "Assign" button

A new panel "Selection" appears. Here the parameter must be selected, on which H 3-4 shall depend. The function "Type 7" can depend on two other parameters (ParA and ParB). In the present case only one of them is needed.

- Mark "P1: H 02-01" in the field "Parameter A" and an arbitrary line (dummy) in the field "Parameter B"

As soon as in both fields a line is marked the "OK" button becomes activ.

- Click the "OK" button

This click activates the assignment of the parameter H 3-4 to the function. Instead of the variable parameter H 3-4 now there are three new parameters (coefficients of the function) with the names P0, PA and PB. By default they have starting values of 1.0 and their status is "variable". Starting values and status can be changed by clicking on the respective value in the now appearing green window.

- Click on "P0 = 1.00", change value to 0.00
- Click on "PB = 1.00", change value to 0.00
- Click on "variable" (P0) → status changes to "fixed"
- Click on "variable" (PA) → status changes to "fixed"
- Click on "variable" (PB) → status changes to "fixed"

The choice of PB = 0 (fixed) causes that "Parameter B" is without influence (dummy).

If functions "Type 1", "Type 2", "Type 3" or "Type 6" are used the computation time will be extended significantly. In these cases the parameters become time dependent, and this makes the calculations much more complicated.


# Special functions

## Daily pattern

This function is useful when analysing data from measurements at a window or glazing.
It is very difficult to measure surface temperatures of glazings exposed to solar radiation. Therefore, only the data from the night time can be used for the analysis. So a lot of fitting windows have to be applied, one per night. To

simplify this the "Daily pattern" function (on the "Output graphics" page) creates automatically the necessary fitting windows.
However, use of fitting windows with a duration of some hours is only possible in thermal systems with a low time constant (significant lower than the duration of the fitting windows).

## Error propagation

This function allows to estimate how systematic errors in the measured data affect the error in the final results. The errors (uncertainties) of the measured data can be specified in the data file, in the line with the column names. Example:

"Time" "Heat" 3% "STI" "ATE" 1% "ATI" 0.5 "GVE" 3 "Tiles" 2%

Explanation: Error for Heat is 3%, for STI there is no error specified, error for ATE is 1%, error for ATI is 0.5 K, error for GVE is 3 W, error for tiles is 2%. Every number behind a column name is considered as an error. If the number is followed by the %-sign, the error is considered as relative, otherwise as absolute.
All of these specifications can be changed in the table on the "Error propagation" page.

## Prediction error method

By using this option a statistical method for finding the parameters is used. For that purpose it is necessary to specify model errors at the nodes and errors at the measured outputs (standard deviations). These errors can be considered as parameters and identified during the identification process.
In general, the calculation time is remarkable extended by using the Prediction error method.


# General hints and remarks


- Try to use different models
- Try to use different starting values
- Try to use different fitting windows
- Use a fast computer and allow the identification to run overnight

During the identification process millions of single operations must be performed. In some (hopefully seldom) cases it is possible that results of

operations exceed certain internal limits and that the program crashes. Of course it would be possible to prevent this and to check each operation. But this would entail longer computation times. Therefore, a certain compromise between computation time and "crash safety" was taken.
Especially when parameter functions are used all inputs should be checked carefully.

In cases of problems, questions, suggestions, comments etc please send an e-mail, at best with the attached data and model file.

olaf.gutschker@tu-cottbus.de