Cervenka Consulting s.r.o.
Na Hrebenkach 55
150 00 Prague
Czech Republic
Phone: +420 220 610 018
E-mail: cervenka@cervenka.cz
Web: http://www.cervenka.cz

SARA Program Documentation

SARA Science

RLACS STUDIO

Structural Analysis and Reliability Assessment

User's Manual

Written by

Petr Havlásek and Radomír Pukl

Prague, 6. 10. 2017

CONTENTS

1 INTRODUCTION

The programs FREET and ATENA are integrated in the software package RLACS (Structural Analysis and Reliability Assessment) in order to allow for a probabilistic nonlinear analysis of concrete structures.

The interactive graphical shell RLACS Studio was developed to assure well-arranged data exchange and management as well as control of both mentioned programs and additional supporting tools. The whole process of the nonlinear stochastic simulation is controlled by the user using commands and interfaces available in RLACS Studio.

1.1 Preparation of deterministic input

The deterministic finite element (FE) model of the analyzed problem should be developed and well tested in deterministic version of ATENA software before the random study is started. All the calculation parameters (load steps, monitoring points etc.) should be prepared for the stochastic calculation in advance and included in this model.

1.2 Randomization of input variables

The material properties and other input parameters used in ATENA deterministic analysis are defined as first. These values are exported to FREET, where they will be used as mean values for random distributions of the corresponding variables. Further stochastic parameters (variance, type of the probability density function) for selected variables are defined directly in FREET. The randomness of input variables reflects uncertainties and randomness of the input values regarding material properties, geometry of the structure, prestressing etc. Integrated database of stochastic parameters for various structural and material properties (concrete, reinforcing steel, prestressing, geometrical imperfections) is available to support the user in preparing stochastic input data. Correlation between random input variables can be introduced in form of the correlation matrix. Number of samples should be selected according to complexity of the problem to be solved and required quality of expected results. Already 8 samples could give a reasonable estimation of stochastic parameters of the structural response and reliability index prediction.

1.3 Repeated nonlinear solution

In the next step, sets of input parameters for the required number of samples are generated by **FREET**. **RLACS Studio** prepares input data for the multiple analysis using **AtenaWin**. The samples are consequently solved in **AtenaWin** under RLACS Studio control. Selected results from the structural response from AtenaWin solution (ultimate load, deflection, maximum crack width etc.) are collected. Finally, obtained results are transferred to **FREET** and evaluated in form of histograms of structural response and sensitivity plots. Reliability index can be assessed.

1.4 RLACS working scheme

The solution procedure can be itemized as follows:

> Create and check deterministic model of the structure in ATENA 2D Engineering

⇩

> Uncertainties and randomness of the input parameters are modeled as random variables described by their probability density functions (PDF). The result of this step is the set of input parameters for ATENA computational model – random variables described by mean value, variance and other statistical parameters (generally by PDF).

⇩

> Random input parameters are generated according to their PDF using Latin hypercube sampling (LHS). Statistical correlation among the parameters is imposed using simulated annealing.

Loop over all samples

⇩

> Generated samples of random parameters are used as inputs for ATENA computational model. The nonlinear solution is performed and selected results (structural response) are saved.

⇩

> The resulting set of structural responses from the whole simulation process is statistically evaluated. The results are: histogram, mean value, variance, coefficient of skewness, empirical cumulative probability density function of structural response, sensitivity evaluation, and reliability index assessment.

2 EXAMPLE OF USING RLACS STUDIO

2.1 Purpose of this example

This example gives a step-by-step instructions how to
- create a new project and how to set its settings;
- select materials, material parameters and/or geometry entities for randomization;
- define properties of material parameters using pdf;
- define correlation among parameters using correlation matrix;
- create randomized inputs and to run FE analyses;
- assess and evaluate results.

This example is recommended mainly for RLACS Studio and ATENA beginners; more advanced users can skip this part and proceed with following chapters.
Please note, that this tutorial uses unmodified ATENA Engineering 2D tutorial input file, which can be found in directory (or similar)
C:\Program Files\CervenkaConsulting\AtenaV4\Examples\ATENA Engineering\Tutorial\Beam iso.cc2
This file will serve as a source for generation of the *.inp file. This input file Beam_iso.inp can be either created using simple procedure (to be explained in section 3.3) or can be found in directory Examples.

2.2 Description of the deterministic problem and response of the structure to loading

Geometry, loading and material properties of this example correspond to the experimental setup used by Leonhard in 1962. Figure 1 shows geometry and loading of this structure; the problem is symmetrical about its vertical axis, hence, only one half is defined in the input file. Symmetry is imposed by the boundary conditions at the axis of symmetry.

Leonhard's simply supported beam is reinforced only with mild rebars located at the bottom. Neither stirrups nor other types of reinforcement are used. The beam is loaded by two downward forces until failure.
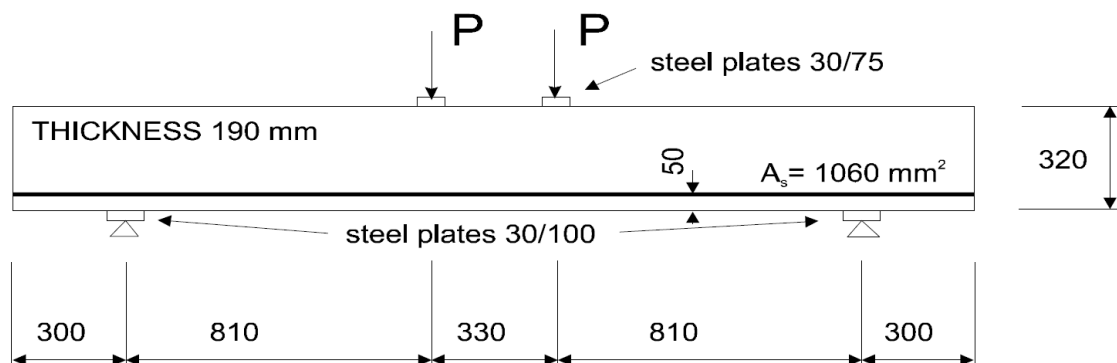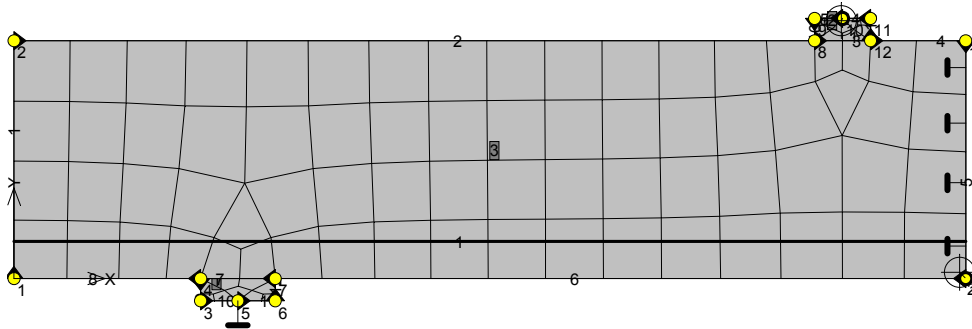

**Figure 1 Geometry and loading of Leonhard's beam**

This example uses quite coarse finite element mesh (see Figure 2) in order to speed up calculations and to enable usage of ATENA demo version. To obtain more accurate results, it is vital to refine FE mesh.

Figure 2 FE mesh, monitoring points and boundary conditions.

There are two monitoring points; the first one measures loading force in the middle of the top loading plate and the second one, located near the axis of symmetry, captures maximum vertical displacement. Since both coefficients multiplying results in these monitoring points are equal to one, all data are negative (vertical y axis is positive upwards). Therefore the default LD diagram is in the 3rd quadrant. The non-linear FE simulation uses 50 incremental analysis steps solved by modified Newton-Raphson method.

Figure 4 shows distribution and orientation of cracks, normal stress in the reinforcing bars (303 MPa) and horizontal normal stress at the peak of the LD diagram = 43. analysis step. Figure 5 shows iso-areas of crack widths for the same solution step. It is obvious, that the failure of this structure is caused by development of a diagonal shear crack.
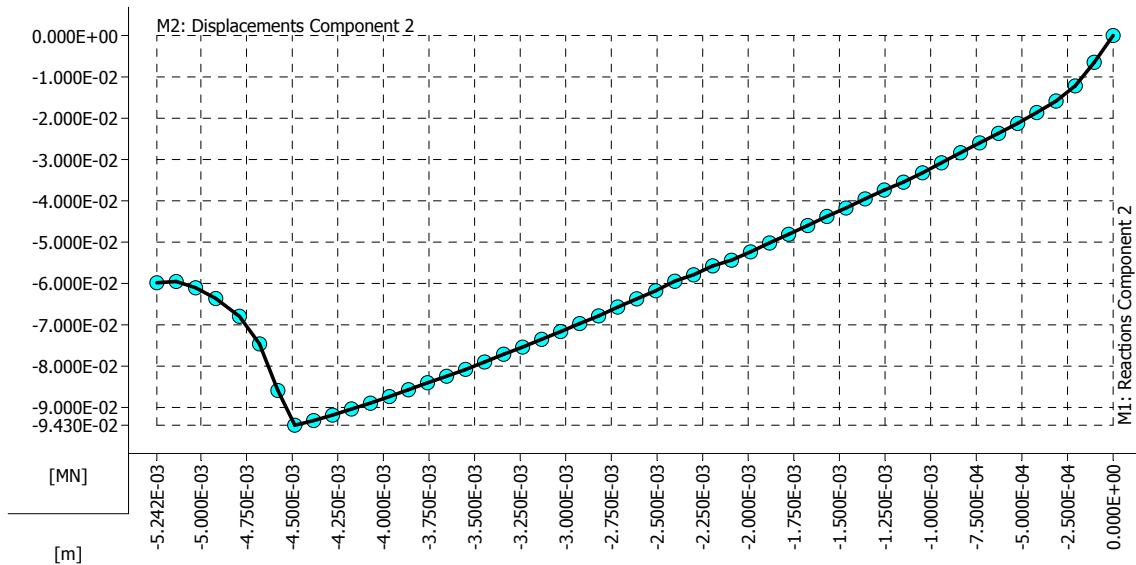
Figure 3 LD diagram

Step 43, Reinforced beam
Scalars:iso-areas, Basic material, in nodes, Stress, Sigma xx, <-4.666E+01;6.887E+00>[MPa]
Cracks: in elements, openning: <5.225E-07;4.685E-04>[m], Sigma_n: <0.000E+00;1.285E+00>[MPa], Sigma_T: <-1.526E+00;3.299E
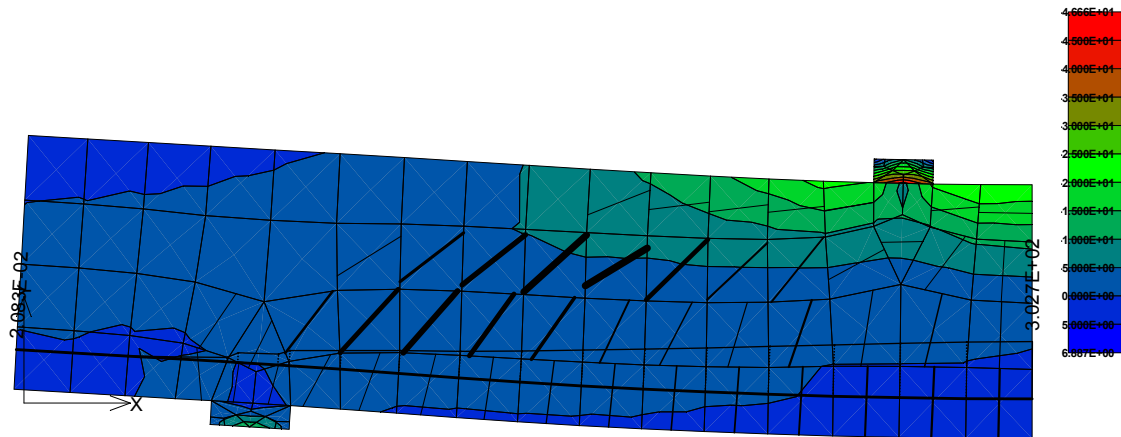Reinforcements: Principal Stress, Max., <2.083E-02;3.027E+02>[MPa]

Figure 4 43. analysis step (peak): cracks, tensile stress in the rebar and iso-areas of the horizontal normal stress.

Step 43, Reinforced beam
Scalars:iso-areas, Basic material, in nodes, Crack Width, Cod1, <0.000E+00;3.649E-04>[m]
Cracks: in elements, openning: <5.225E-07;4.685E-04>[m], Sigma_n: <0.000E+00;1.285E+00>[MPa], Sigma_T: <-1.526E+00;3.299E
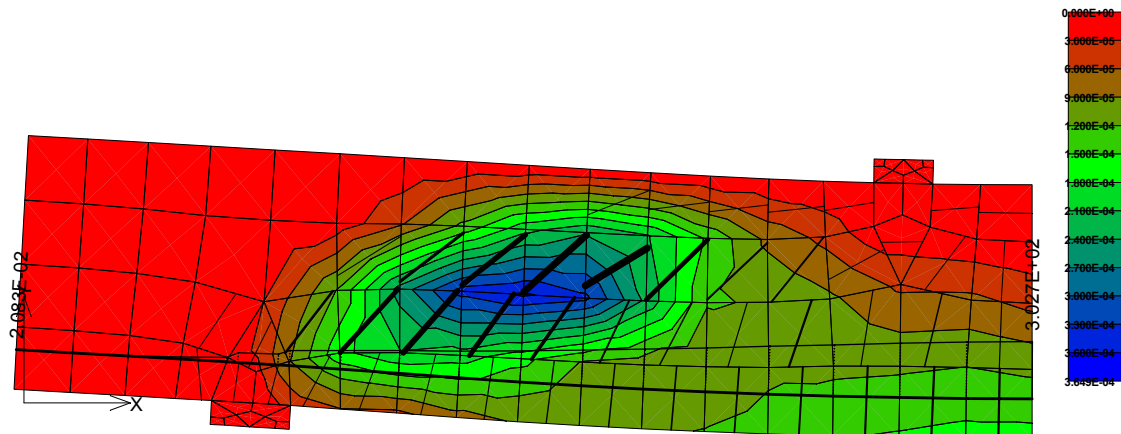Reinforcements: Principal Stress, Max., <2.083E-02;3.027E+02>[MPa]

Figure 5 43. analysis step (peak): iso-areas of crack width (max. 0,36 mm)

In this RLACS Studio example it is assumed that the geometry of the specimen and the position of loading plates, forces and supports are 100% accurate, i.e. it is considered to deterministic; the source of randomness originates only from material properties.

2.3 Starting RLACS Studio

First of all, the program RLACS Studio needs to be started. This can be done either using the shortcut, which was created during ATENA installation, or by following the standard path in Windows start menu: **All Programs|Cervenka Consulting | ATENA Engineering | RLACS Studio** or finally by running directly the RLACS Studio executable file in your ATENA Engineering installation directory (eg. C:\Program Files\CervenkaConsulting\AtenaV4\RLACS.exe). The introductory window shown in Figure 6 should appear. Make sure that the hardware key is properly linked.
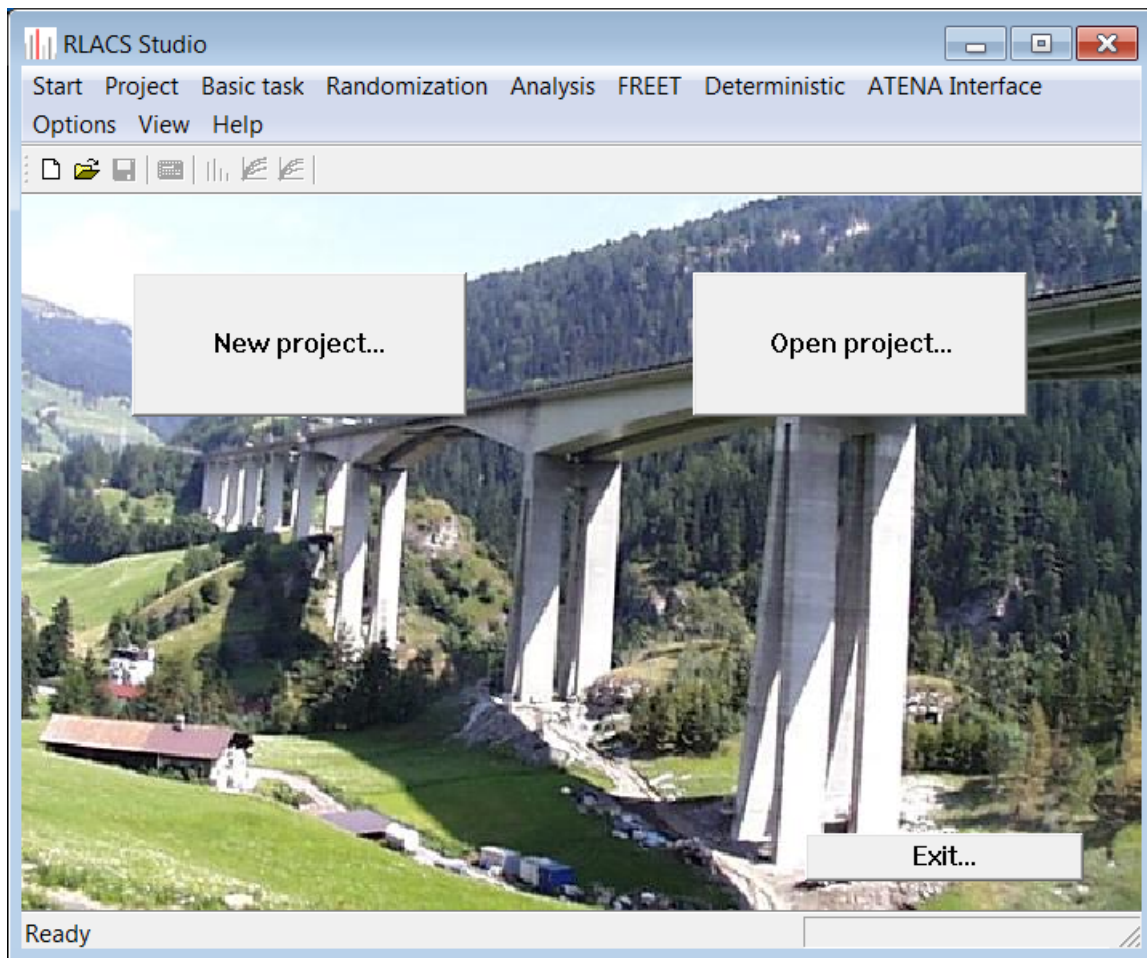
Figure 6 RLACS Studio introductory window

A fully sufficient way of proceeding through the project in RLACS Studio is by following big buttons in RLACS Studio main window. Some of them are hidden in the beginning, but will appear successively. Follow them to guide you through the analysis.

More options and features can be found in the main program bar placed at the top of the main window. All options in this bar are ordered accordingly to expected progress of the user.

The language of ATENA and the working directory (relative or absolute path) can be set choosing **Options │ General**. A following window, shown in Figure 7 appears.

The working directory path can be adjusted any time in order to locate existing projects or to create new project. The working directory can be set either relatively to AtenaV4 installation directory or absolutely. The default working directory is named "Projects", and is located in ATENA installation folder. If the **Relative** option is chosen and the input box is left blank, then the new project will be created directly in the AtenaV4 installation folder (this is not recommended). Note that in case you are using Windows 7 and **RLACS.exe** is located in Program files, RLACS must be "Run as administrator" in order to enable use of the relative path.

Figure 7 shows, that working directory named "RLACS_projects" was chosen absolutely, and that English language can be chosen eventually.
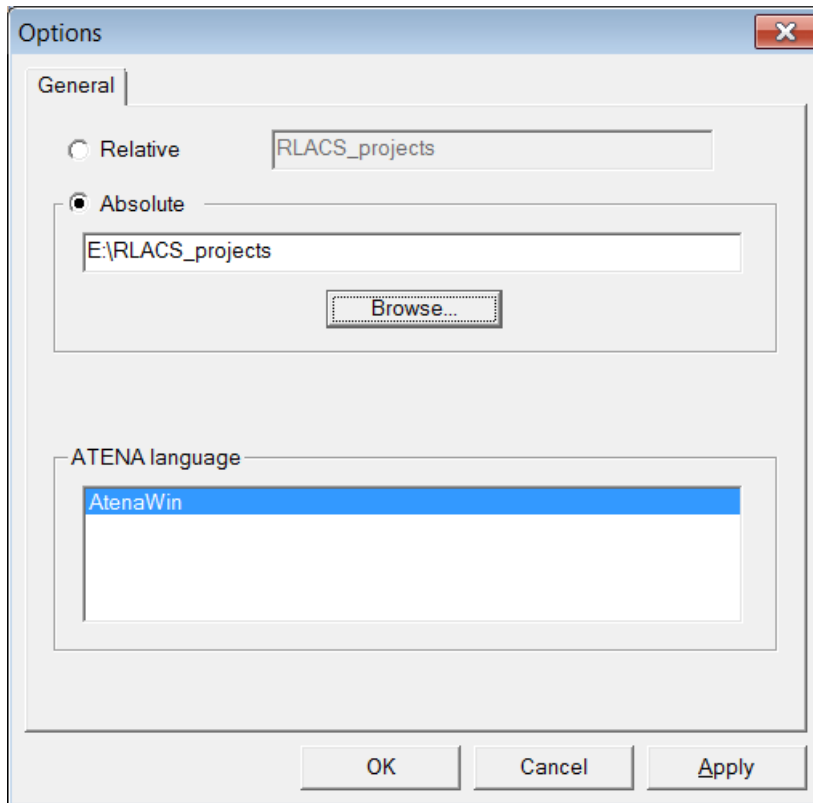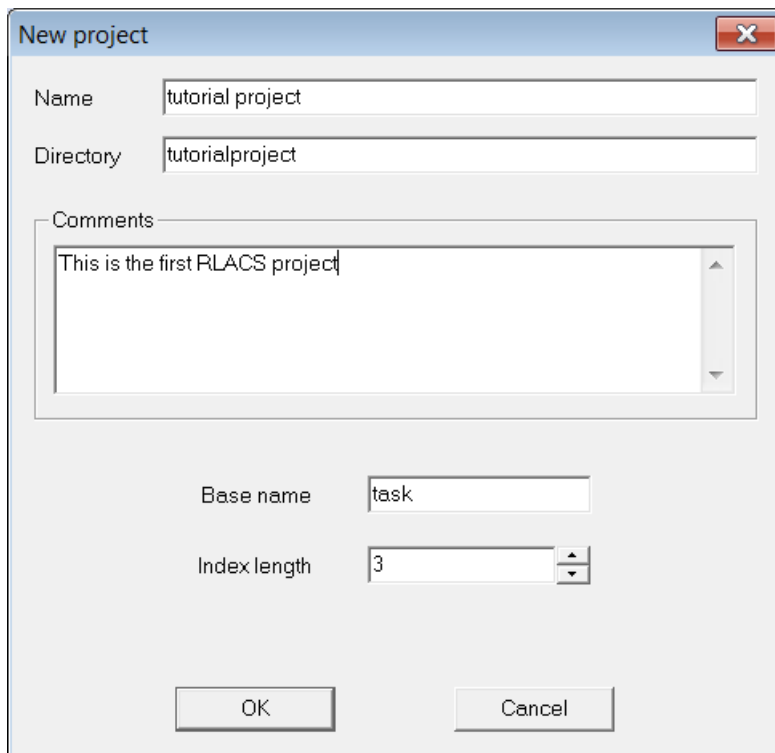
Figure 7 General options windows (Options ⇨ General)

2.4 Creating a new project and importing basic task

There are two ways to create a new project. The first one is using pull-down menu **Project │ New...**, the second possibility is the button **New project...** in the upper left corner of the main window.

Figure 8 New project window

The window shown in Figure 8 displays the **New project** window, where the name of the project, the name of the parent directory, comments, the basic task name, and index length are to be specified.

Name specifies the name of the project

Directory determines the name of the folder, which will be created in the working directory. The name of this directory is created automatically, but can be modified (in our case, the path to this project will be …\AtenaV4\RLACS_Projects\tutorialproject\).

Comments - this field is not compulsory, but can contain details or closer information about the problem

Base name determines the name of the basic task, which will be created in the project folder as a copy of the original, deterministic problem. The default **Base name** is "task". Base name is also the first part of the name of all randomized input files. The Base name is followed by several digits, which define an index number of the particular randomized input. The number of digits is defined by **Index length**. If the index length is equal to 3 (default value), then it is possible to create up to 999 randomized input files, which is usually fully sufficient.

Finally, click **OK** to confirm your input.
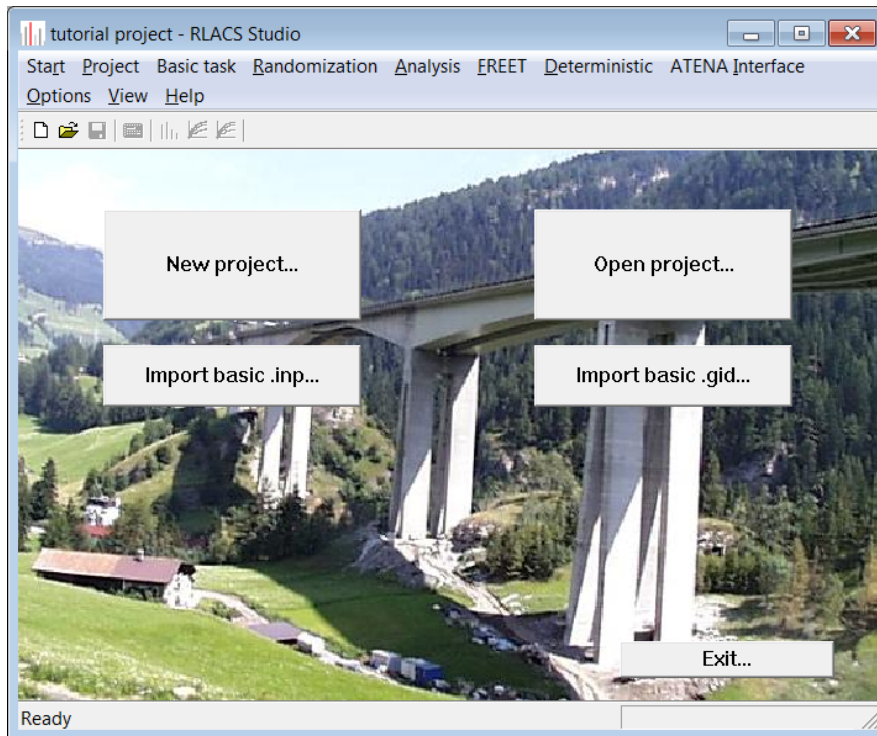
Figure 9 The main window just before importing the basic (deterministic) problem

When the project is created, a button labeled **Import basic .inp …** appears in the RLACS Studio main window. In the following step, the deterministic basic task is imported. Either click **Import basic .inp …** button, or the same function can be found in the main project bar under **Basic task │ Import .inp …**

In this tutorial example, the four-point bending test of a reinforced concrete beam was chosen as the basic task. This project can be created according to ATENA Engineering 2D tutorial and afterwards exported to *.inp file, or it can be found directly on the following path:

AtenaV4\Examples\ATENA Science\Tutorial\Beam Iso

How to create a *.inp file from **ATENA 2D** (from *.cc2 file) is explained in detail in section 3.3.
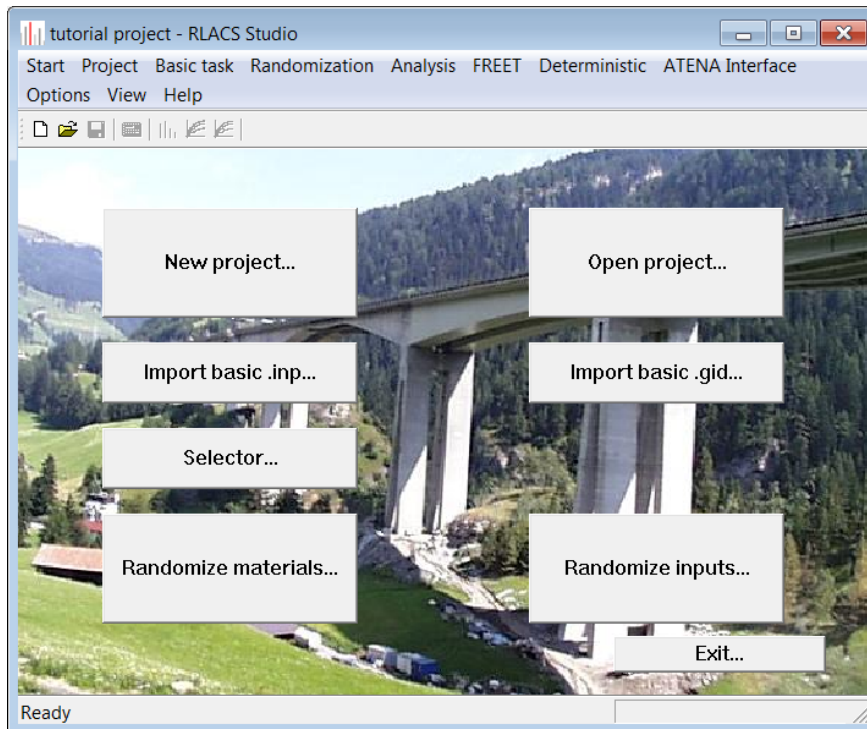
Figure 10 The main RLACS Studio window, once the basic task has been imported.

After the basic task has been chosen and imported (this operation may take up several minutes, depending on complexity of problem and the number of analysis steps), several windows appeared in the main RLACS Studio window, see Figure 10.

In the next (optional) step, it is possible to verify, whether the correct deterministic (mean) basic task has been imported. To do that, in the program ribbon click **Basic task │ Open basic .gid**. The program **GiD** and the basic task is then opened automatically. If necessary, the basic task can be also modified. To apply changes in the basic task, the project must be saved.

2.5 Selection and randomization of input parameters

Once the deterministic problem is prepared, the probabilistic analysis can start. First, it is necessary to choose the scope of the probabilistic study. In this example, only some of the parameters of concrete material model are to be randomized.

Click the button **Selector...** (or use the main program bar **Randomization | Selector...**) and a dialog appears as shown in Figure 11 which offers a choice of parameters, which one intends to randomize. As you can see, no materials are displayed. That is because no materials or material parameters have been marked for randomization in the input file.

Now, it is necessary to close Selector (click **Cancel**), modify input file and to import it once again.
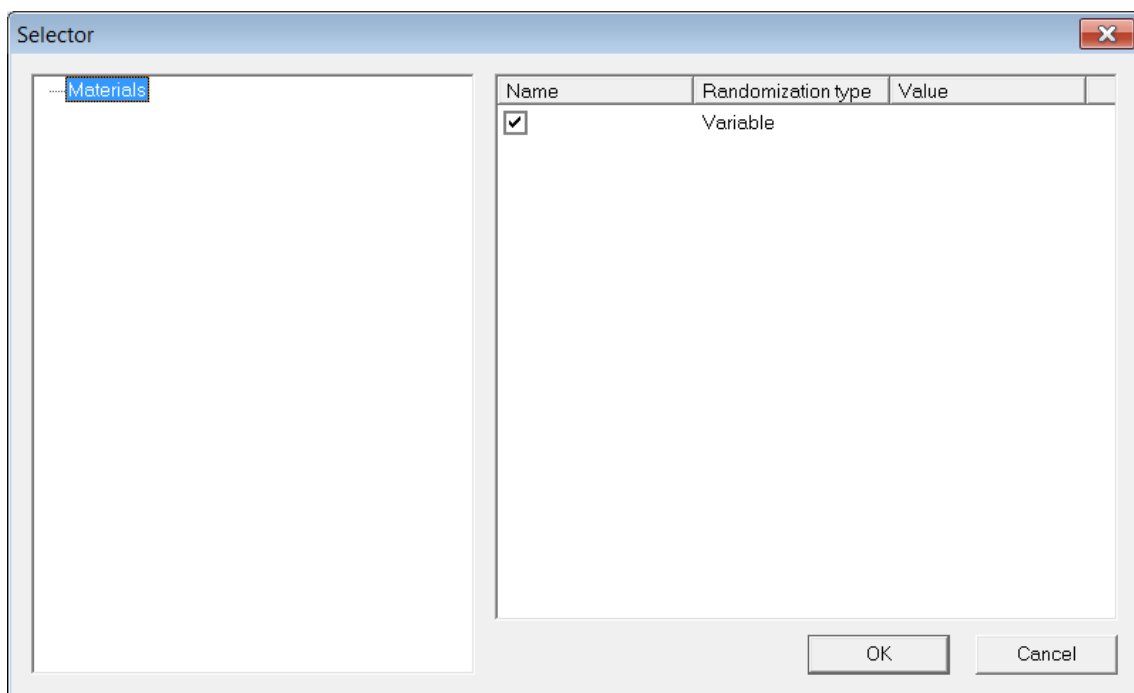
Figure 11 Selector dialog – initial view when nothing has been selected for randomization

See section 3.5 for explanation how to select parameters in ATENA input file in order to handle them to **Selector**. Modify the input file as shown in the next figure or import Beam_iso_selected.inp, which contains all necessary adjustments.
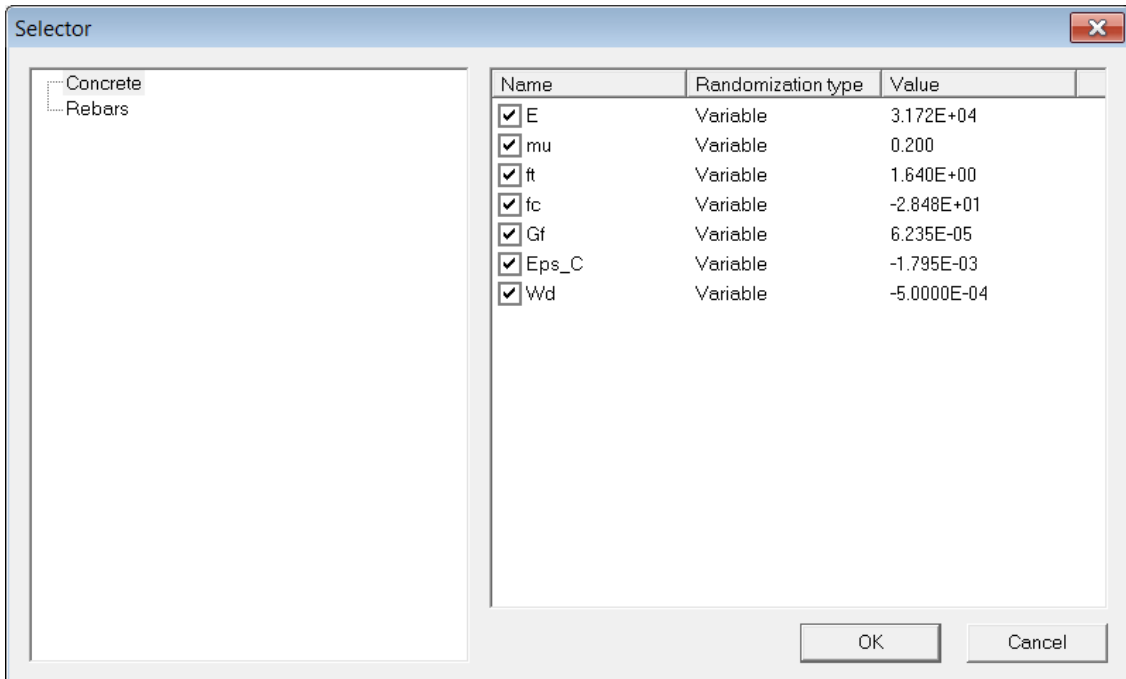
Figure 12 ATENA input file

Figure 13 Selector dialog – initial view: variables of the first material group

Consecutively expand all categories and check parameters, which will be randomized. To efficiently work with long lists of parameters or values, use keys ↑ and ↓ for navigation and space bar for selection.

Only four material parameters of concrete are selected (checked) in this example (Figure 14). One may as well select all parameter, but randomize only some.

Finally, click **OK** to confirm your selection.
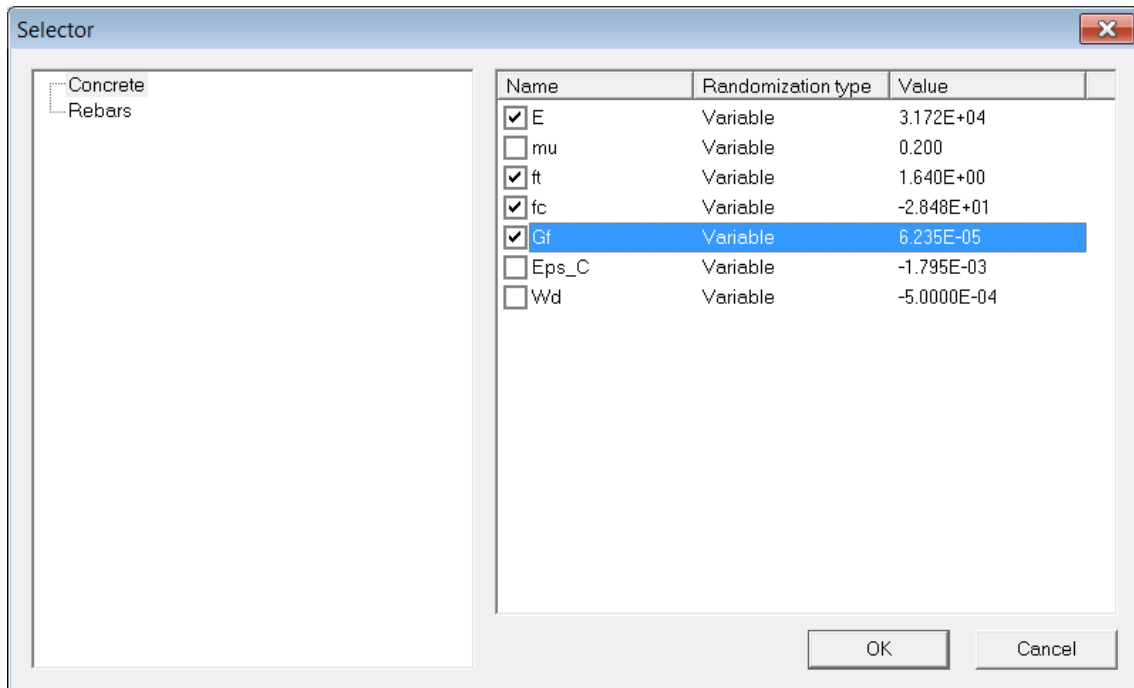
Figure 14 Selector dialog – selection of material parameters for randomization

Click **Randomize materials…** (or use the main program bar and click **Randomization ⎪ Materials⎪Randomization …**) to start the program **Freet**, which is a very handy and powerful tool for generation of random data sets and finally for evaluation of a probabilistic analysis.

In **Freet**, first, for each variable, the PDF is selected, then in the second step, the correlation among variables is prescribed and finally, number of simulations is specified and randomized ATENA inputs are automatically generated.

2.6 Definition of random variables

For all variables, which one wants to randomize, the probability density function (PDF), described by moments, parameters, or both must be defined. For most of the engineering problems, the variables have normal distribution, which can be specified using the mean value and the standard deviation.

To change and to describe PDFs for particular variables, expand **Stochastic model** in the tree menu on the left side of the window and then select **Random variables** (see Figure 15).

The default distribution is the Dirac, i.e. deterministic distribution. This is used when specific variable should have same value in all realizations.

Sometimes, when a variable has a prescribed limit (e.g. tensile strength > 0, compressive strength < 0, ...) a lognormal distribution can be efficiently used.

All default parameters of PDFs are related to the basic task. The mean value is the same as it is in the basic task and the standard deviation is taken as one-tenth of this value. Of course, these parameters are only suggested and can be changed.
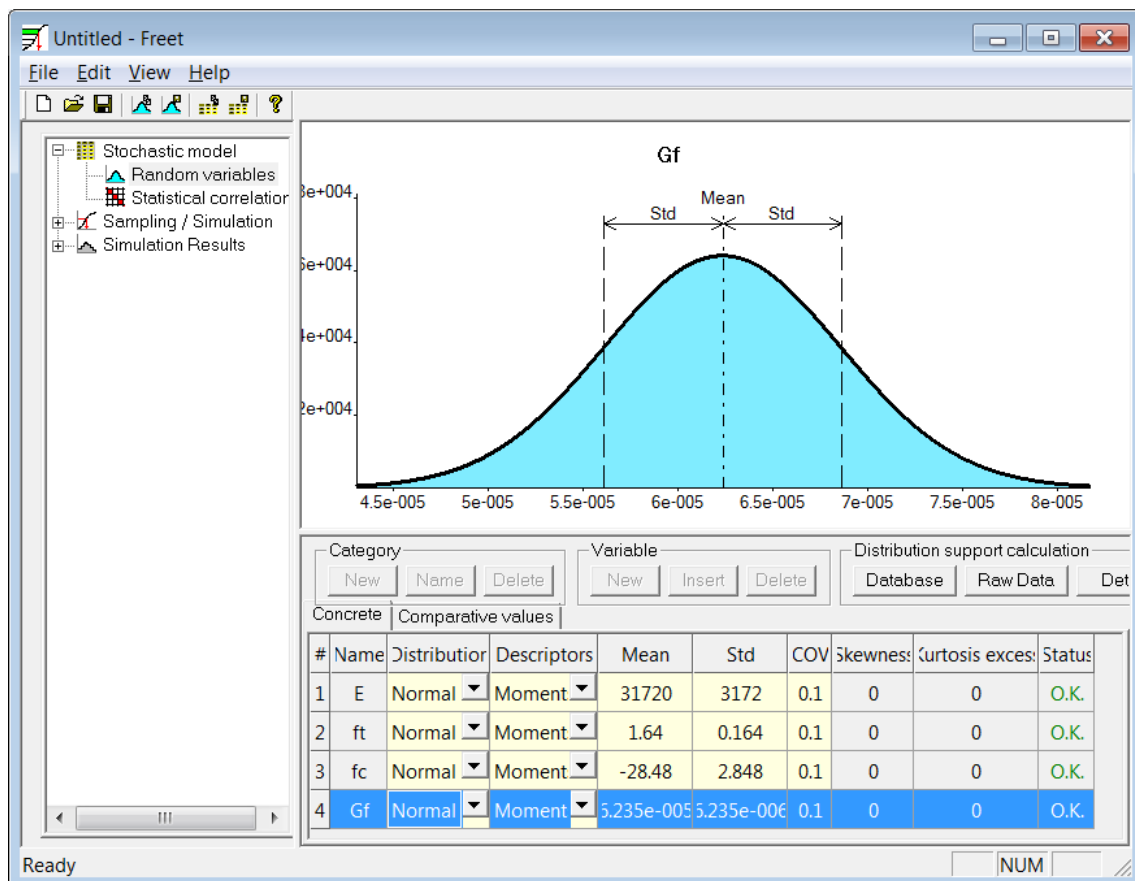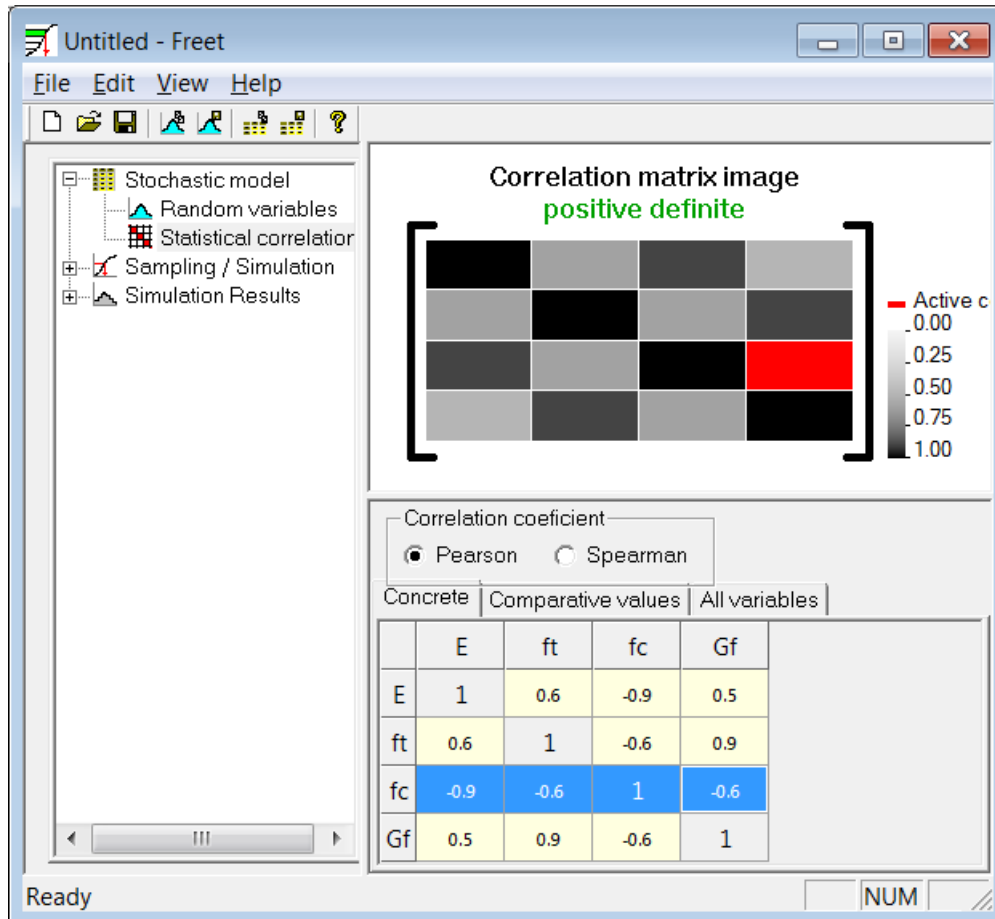

Figure 15 Freet – definition of random variables

2.7 Statistical correlation

To prescribe correlation among variables, expand **Stochastic model** in the tree menu on the left side of the window and then select **Statistical correlation** (see Figure 16).

Figure 16 Freet – definition of statistical correlation

Use numbers ranging from 0 to 1 to prescribe correlation among variables in the correlation matrix. Default correlation matrix has all numbers on the diagonal equal to one (numbers on diagonal are fixed and cannot be changed) and the rest is equal to zero.

Red square indicates currently active cell. In the remaining cells the darker the color, the higher the magnitude of the contents.

The correlation matrix must be symmetric and should be positive definite. If these conditions are met, the samples can be generated using LHS as described in the next section.

Note that if positive correlation among variables of opposite signs is to be imposed, then the correlation coefficient must be negative. One example is the compressive strength, which has a negative mean value, but its magnitude almost linearly depends on Young's modulus – see Figure 16.

2.8 LHS

If all desired PDFs and the correlation matrix are defined, one can proceed to Latin Hypercube Sampling.

Expand **Latin Hypercube Sampling** in the tree menu on the left side of the window and then select **General Data**. To prescribe correlation among variables, expand **Stochastic model** in the tree menu on the left side of the window and then select **Statistical correlation** (Figure 17). Specify **Number of simulations** (=number of nonlinear FE analyses to be performed with modified parameters) and then click **Run** button in the lower right corner of the window to start simulated annealing optimization algorithm. In few seconds, depending on the complexity of problem, specified variables are randomized.

To see the difference between the desired and the actual correlation matrix, click **Check samples**, see Figure 18. The green color (upper right part of the matrix) corresponds to prescribed values and red color (lower left part of the matrix) to values generated using simulated annealing. If the difference between these values is small, one can proceed to the non-linear finite element analysis.

To check generated sets of values of random variables click **Check variables data** (see Figure 19). Each row of the table represents one realization; each column corresponds to one randomized variable. Displayed accuracy of numbers can be changed using dialog box **Digits** located above the table.
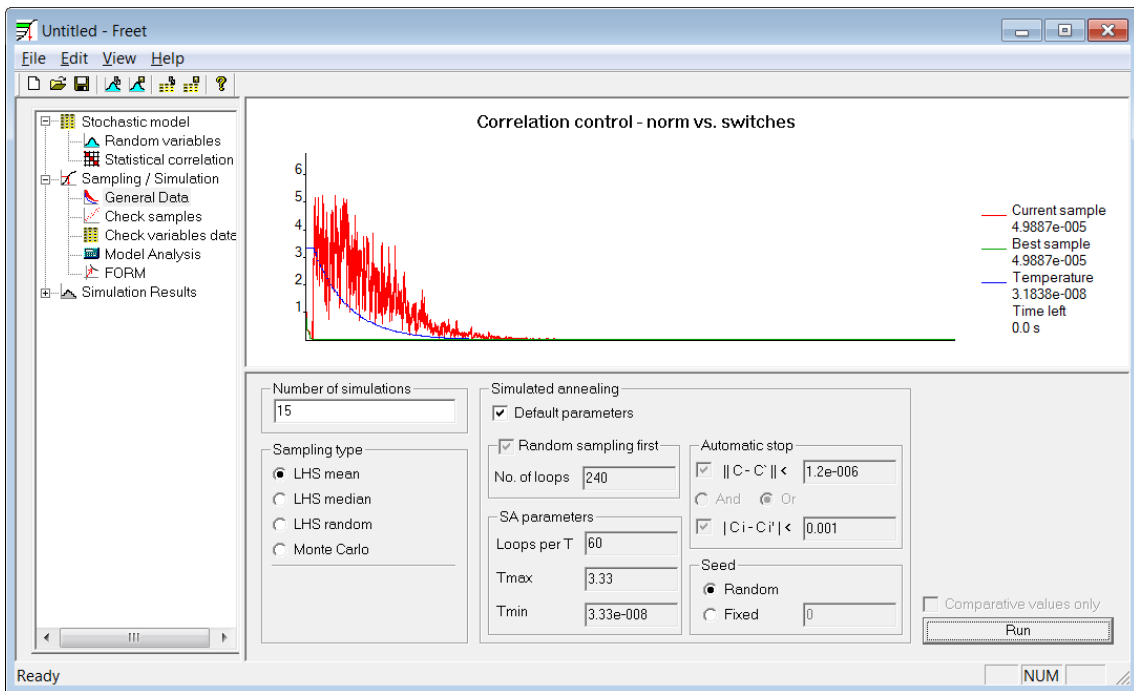
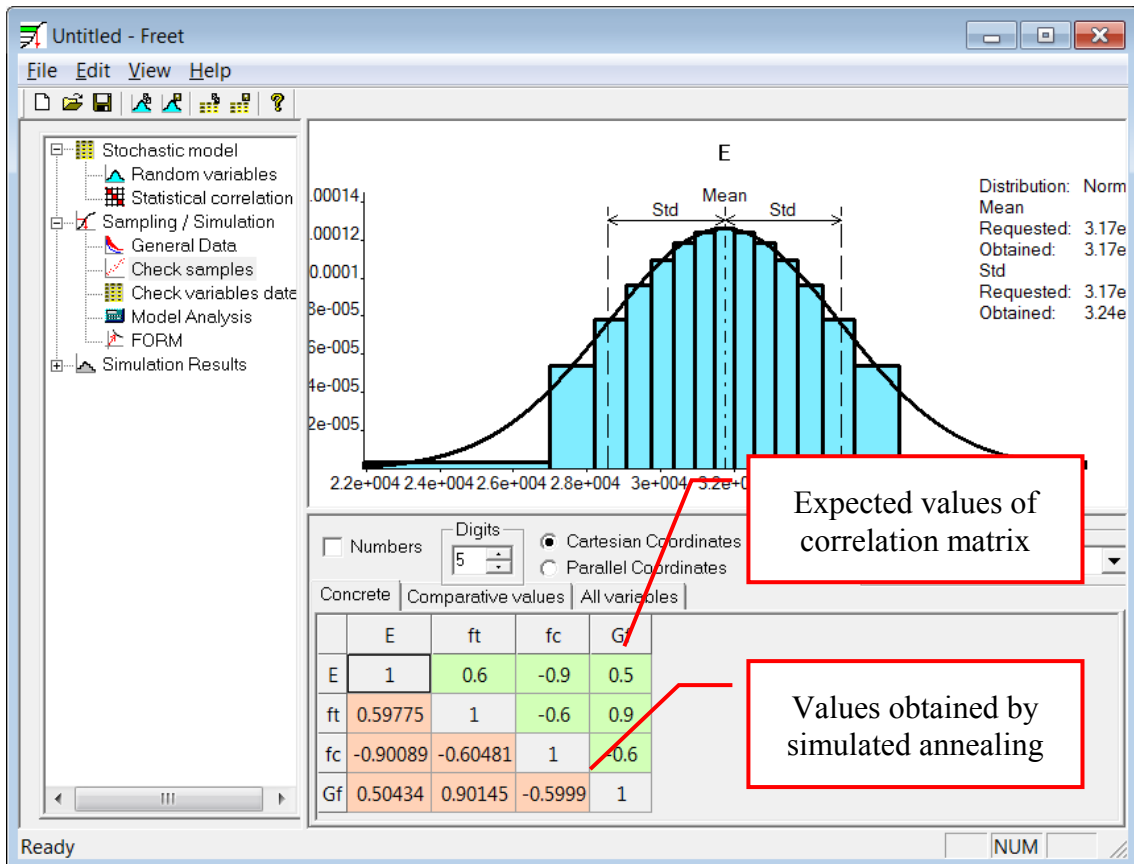**Figure 17 Freet – generation of randomized parameters**

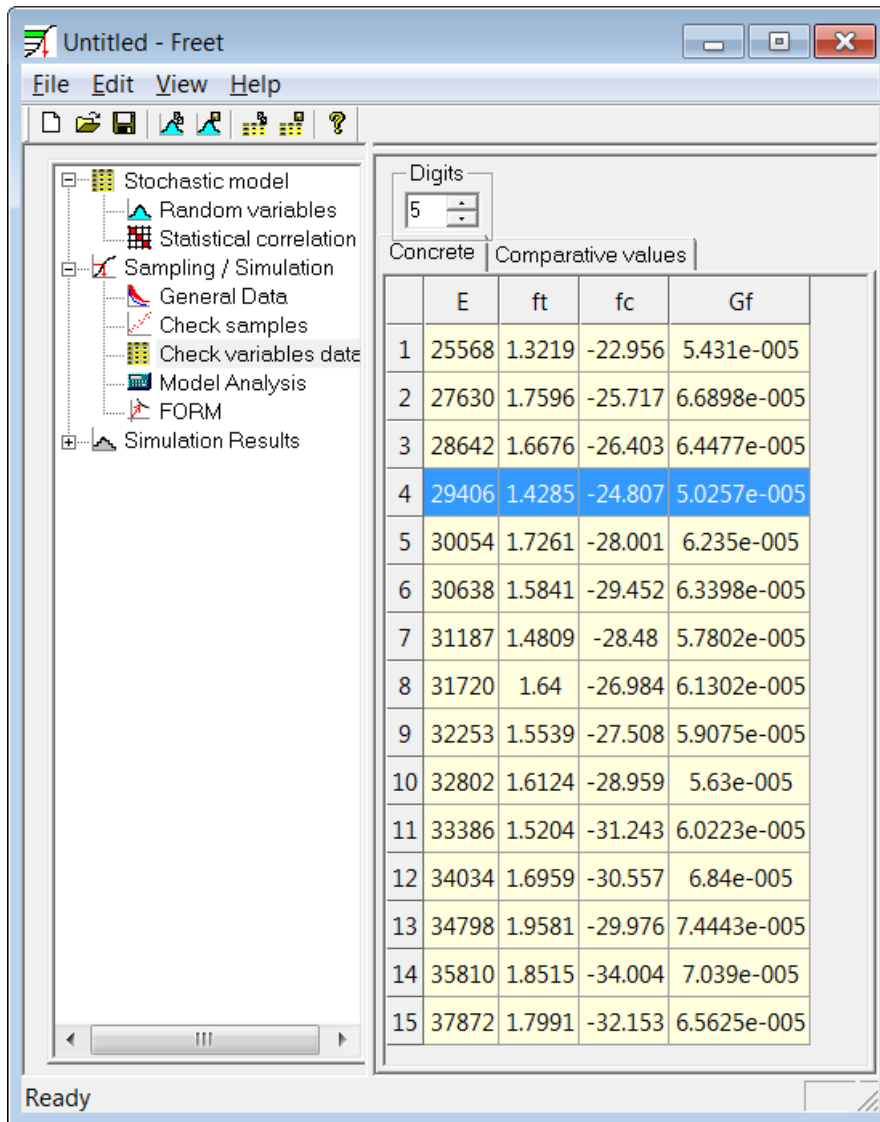**Figure 18 Freet – generated correlation matrix**

Figure 19 Freet – generated randomized sets

2.9 Running FE analysis

After clicking **Model Analysis** in **Latin Hypercube Sampling** group, the set of ATENA input files is generated. This operation can take from few seconds to few minutes depending on the number of simulations and the complexity. In the end, the project folder will contain specified number of folders (in this case 15) named as was described in section 2.4. In this specific case the folders are named task001 … task015. Each one contains files "name of the parent folder".inp and "name of the parent folder".src.

After all tasks have been generated, again RLACS Studio window appears (see Figure 20). The left green column represents a stack of not computed analyses. To start the FE solution in **AtenaWin**, click **Analysis|Run analyses…** or click on the icon with a calculator placed on the top ribbon. Instantaneously, the dialog **Select job** (Figure 21) appears. To run all analyses, click **Select all** and then **OK**.
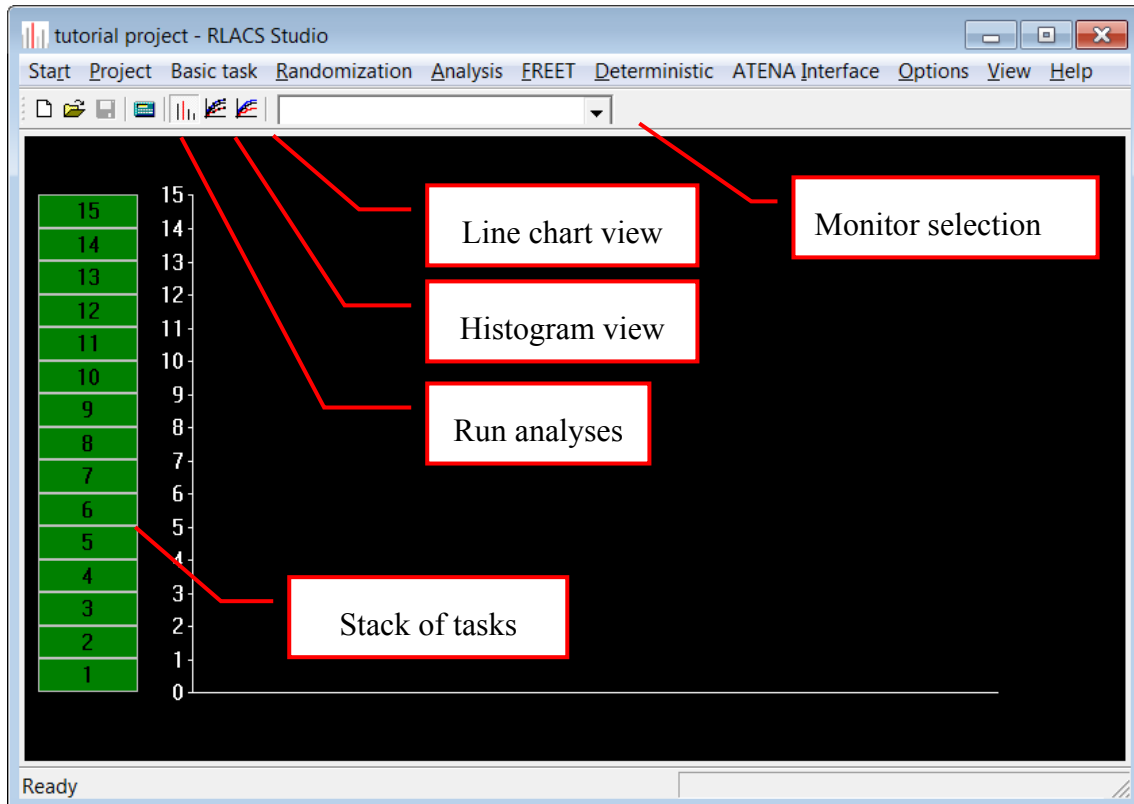
Figure 20 RLACS Studio – stack of inputs ready to be computed
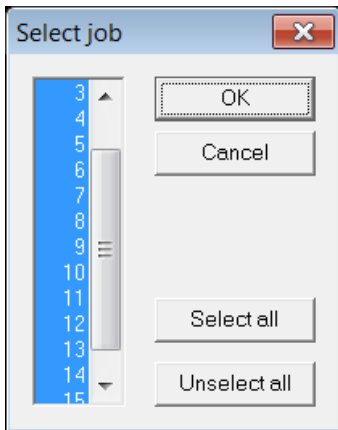
Figure 21 Selection of tasks
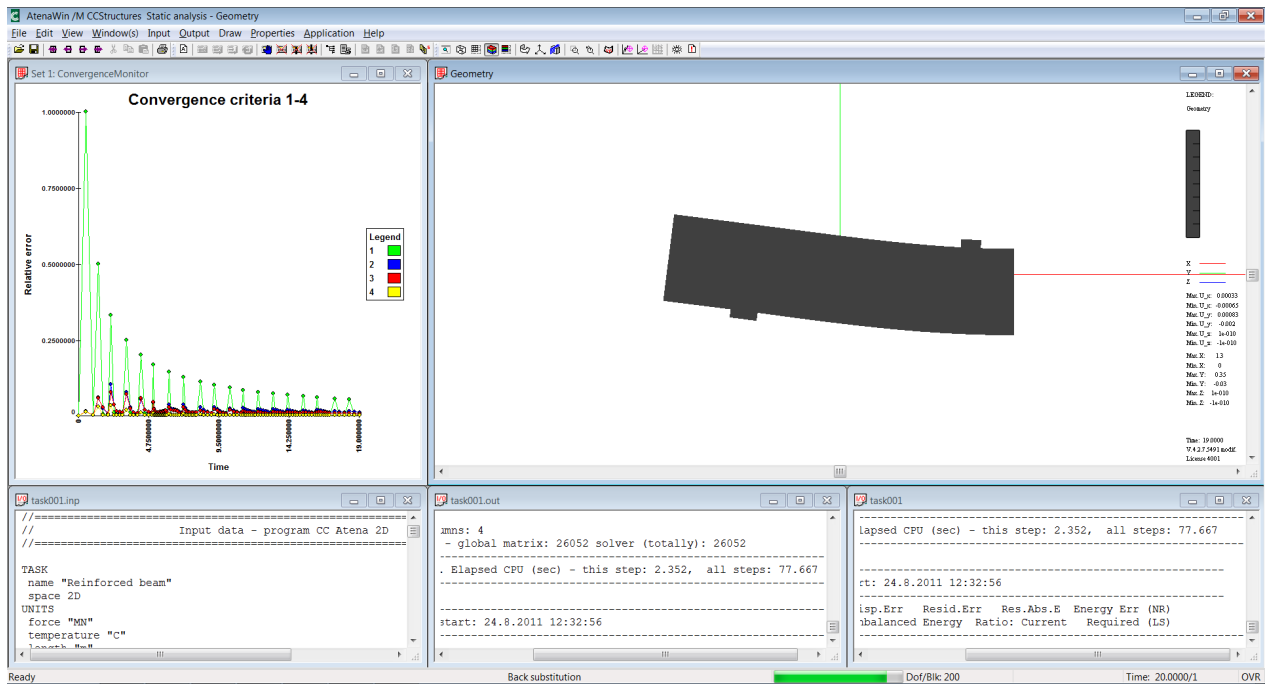
Figure 22 Analysis progress – AtenaWin

Immediately, an AtenaWin window appears (Figure 22) and the solution of the first selected task starts. The user is free to visualize deformed shape, crack width contours, etc. .. When each task is finished, the box with the task number is placed into a graph, according to computed result.

In Figure 23 one can see a typical view of the analysis progress – three tasks have been computed and 15 are left.
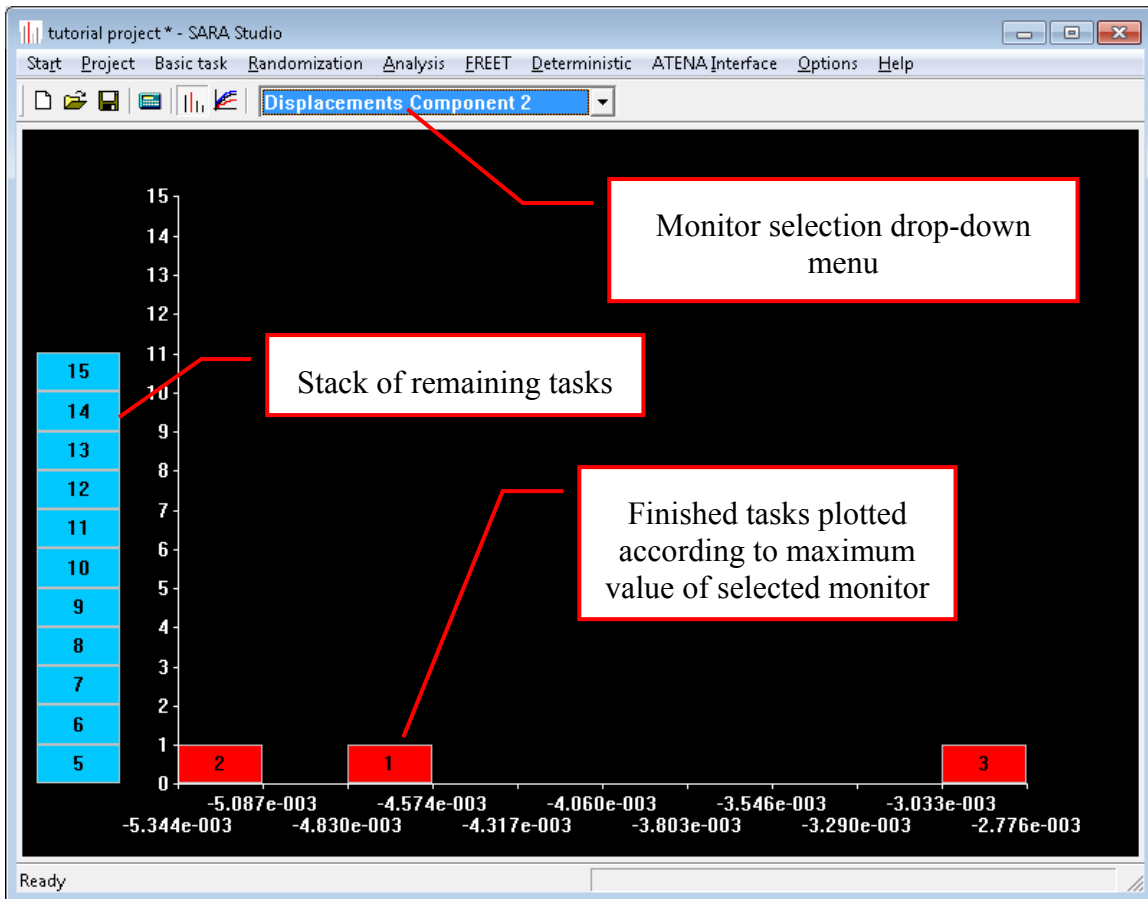
Figure 23 Simulation progress – three finished task and 15 remaining

Figure 24 shows the result of a finished analysis. The computed results form a distribution density according to the **maximum** reached value of Reaction component 2. The distribution density changes according to currently selected monitor, which can be switched at the top of the window. An obvious scatter in the data distribution can indicate, that something went wrong during the analysis and that it might be necessary to modify and recompute some tasks.

Click **Options │ Histogram** to change settings (data ranges, resolution = number of steps, step = step length) of the histogram view. The step length is computed automatically from the data ranges and number of steps.

Figure 24 Finished analysis – distribution of results ordered by "Reactions Component 2"

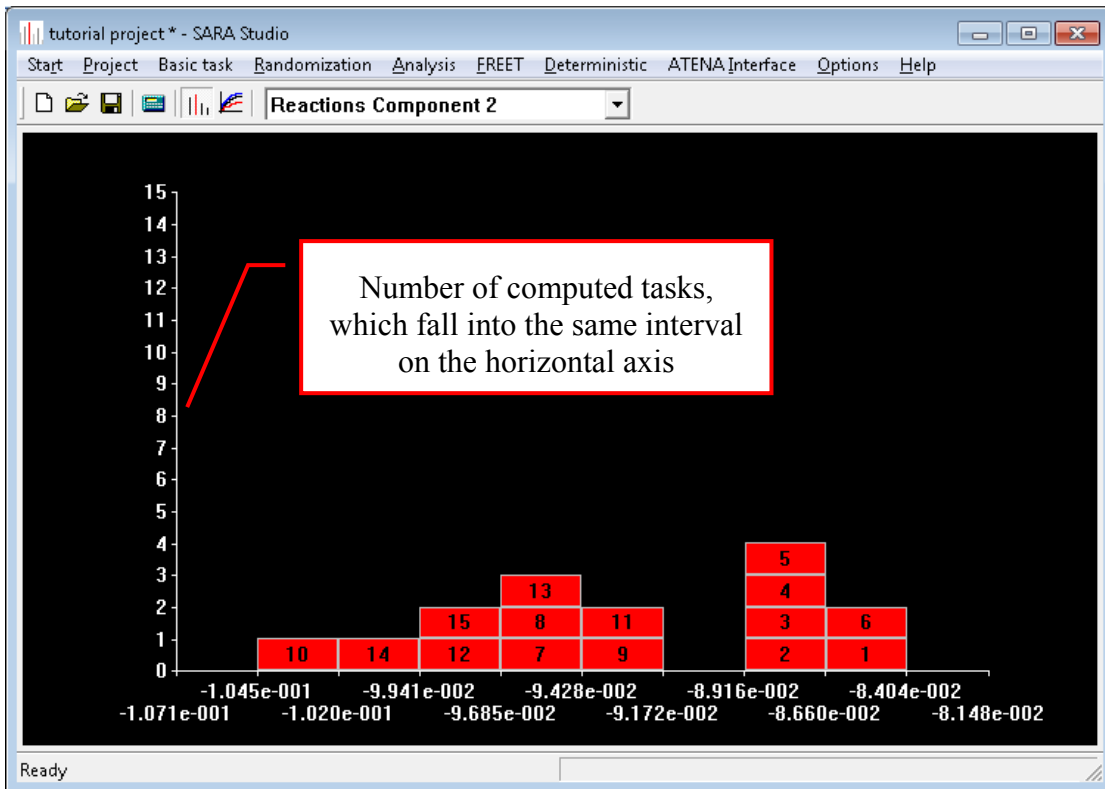This histogram view can be switched any time to a line chart view, showing response to loading of all tasks (load/displacement, displacement/time step, etc …). To do that, click on a **Line chart view** icon as shown in Figure 20.
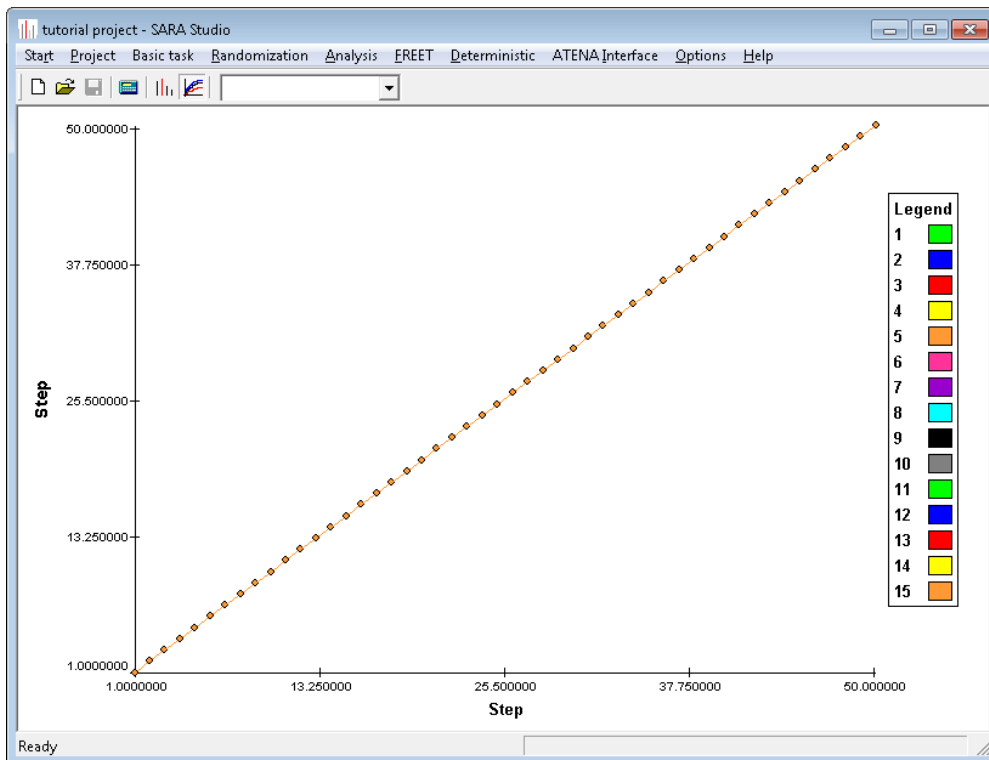
Figure 25 Finished analysis – Line chart view (default view)

In order to show load-displacement response of individual tasks, right-click anywhere in the chart area and dialog appears (Figure 26). Select **Displacements Component 2** for X Axis and **Reaction Component 2** for Y Axis. It is also possible to highlight a particular task - to do that, select it from the drop-down menu at the top of the program window, next to the **Line chart view** icon, see Figure 27.

Unfortunately, all LD diagrams are not properly oriented, because both values of monitor **Displacements Component 2** and **Reactions Component 2** are negative. To transform this graph to the first quadrant, it is necessary to modify the original task in **ATENA**. (Click **Basic task │ Open basic .gid**, multiply all monitors by -1, save the task and run all analyses once again.)
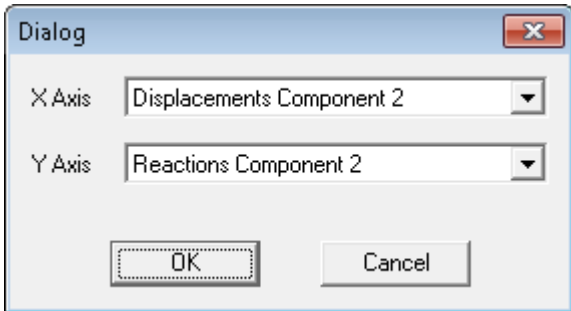
Figure 26 Selection of monitors for horizontal and vertical axis
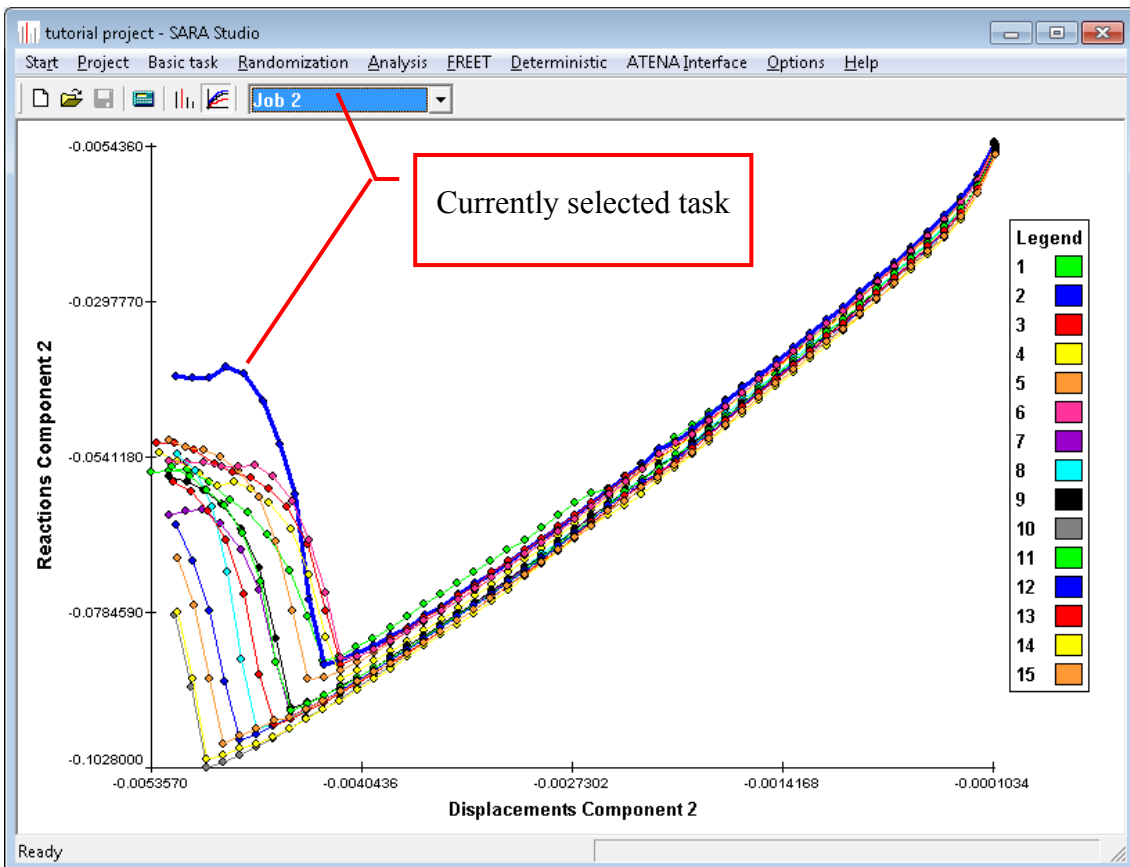
Figure 27 Finished analysis – Line chart view

2.10 Evaluation

To make use of computed results, it is advisable to use **Freet**. To run **Freet**, in the main menu click **FREET ⇨ Statistic Evaluation**. The RLACS Studio window remains open, but becomes inactive. Expand **Simulation Results Assessment** in the tree menu in the left and then select **Histograms** (Figure 28). A concise table shows the main statistical characteristics for each monitor. To evaluate sensitivity, click **Sensitivity analysis** in the same menu (Figure 29 and Figure 30 show positive sensitivity of **Reaction component 2** on absolute value of compressive strength fc). Figure 31 shows positive sensitivity of Reaction component 2 on Young's modulus E.
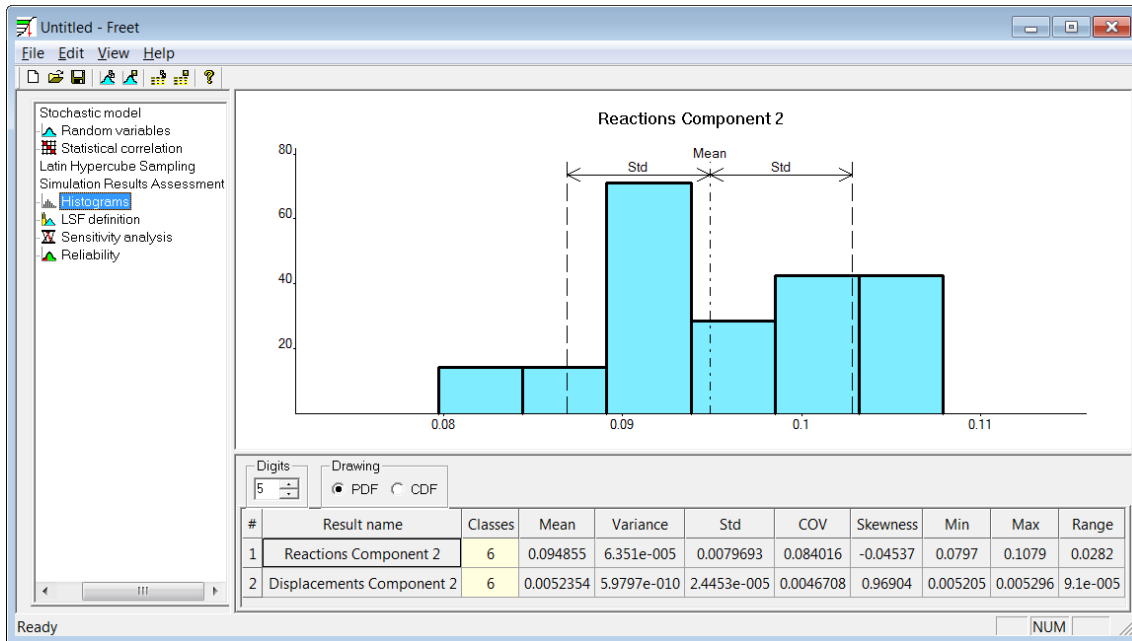
Figure 28 FREET – histogram view

At the end of evaluation, save your project both in **FREET** and **RLACS**: **File | Save** and close FREET, **Project | Save** and exit **RLACS Studio**.
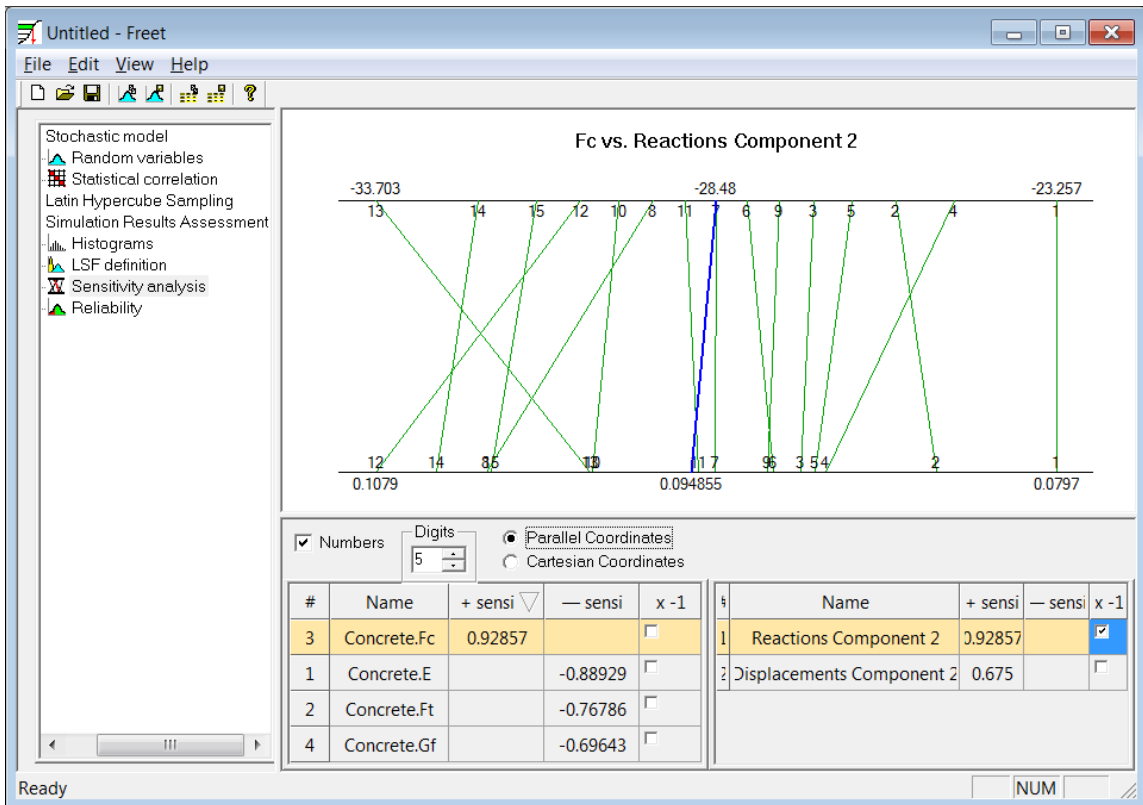
Figure 29 FREET – Sensitivity analysis (parallel coordinates)
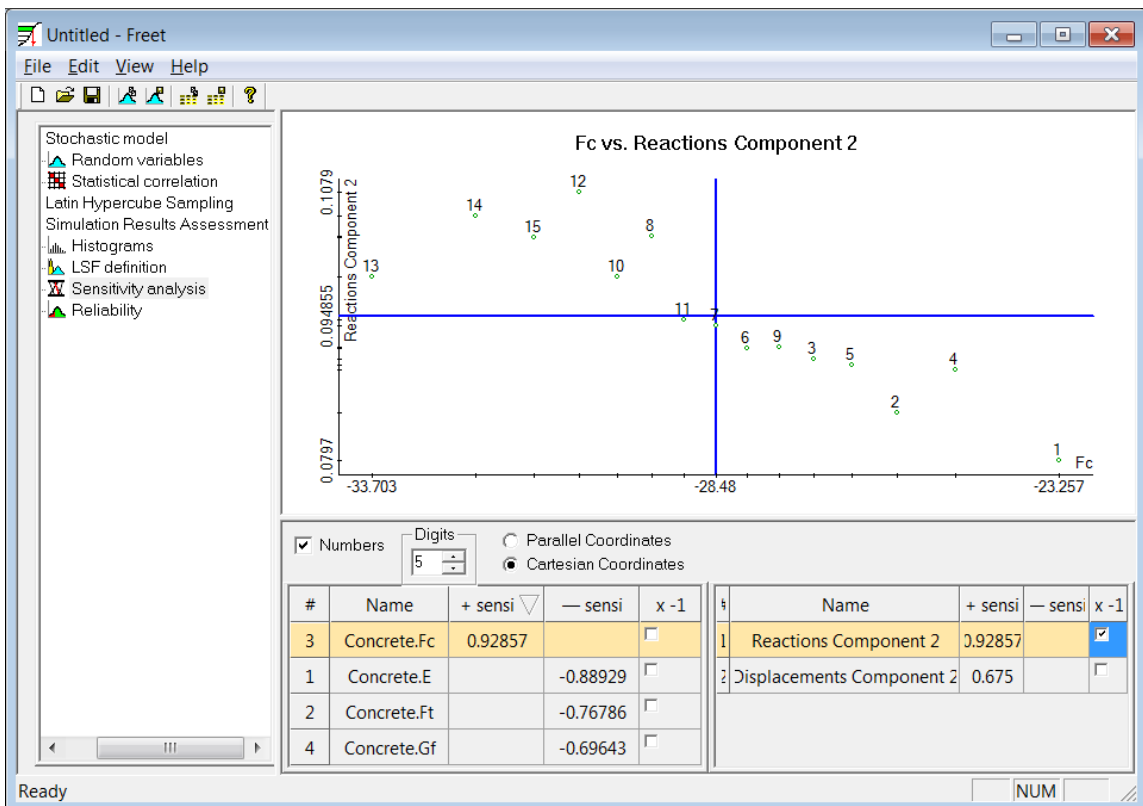
Figure 30 FREET – Sensitivity analysis of Reaction component 2 on compressive strength fc (Cartesian coordinates)
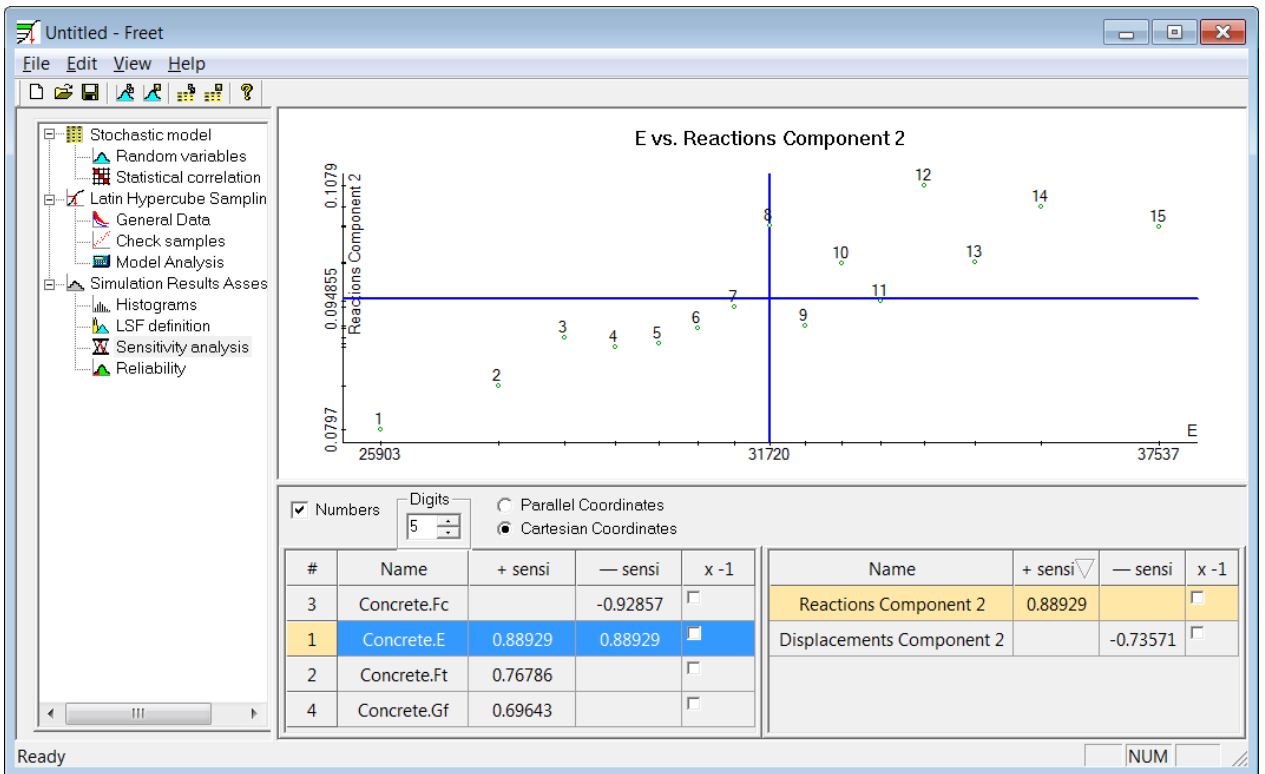
Figure 31 FREET – Sensitivity analysis of Reaction component 2 on Young's modulus E (Cartesian coordinates)

If one knows probabilistic description of acting load, one can determine probability of failure and the reliability index β. Let's assume, that this beam should resist force, which is described by normal distribution with mean value 0.06 and COV 0.1. Go to **Stochastic model** in the tree menu and select **Random variables**. Choose tab **Comparative values** and define new variable named "Action".
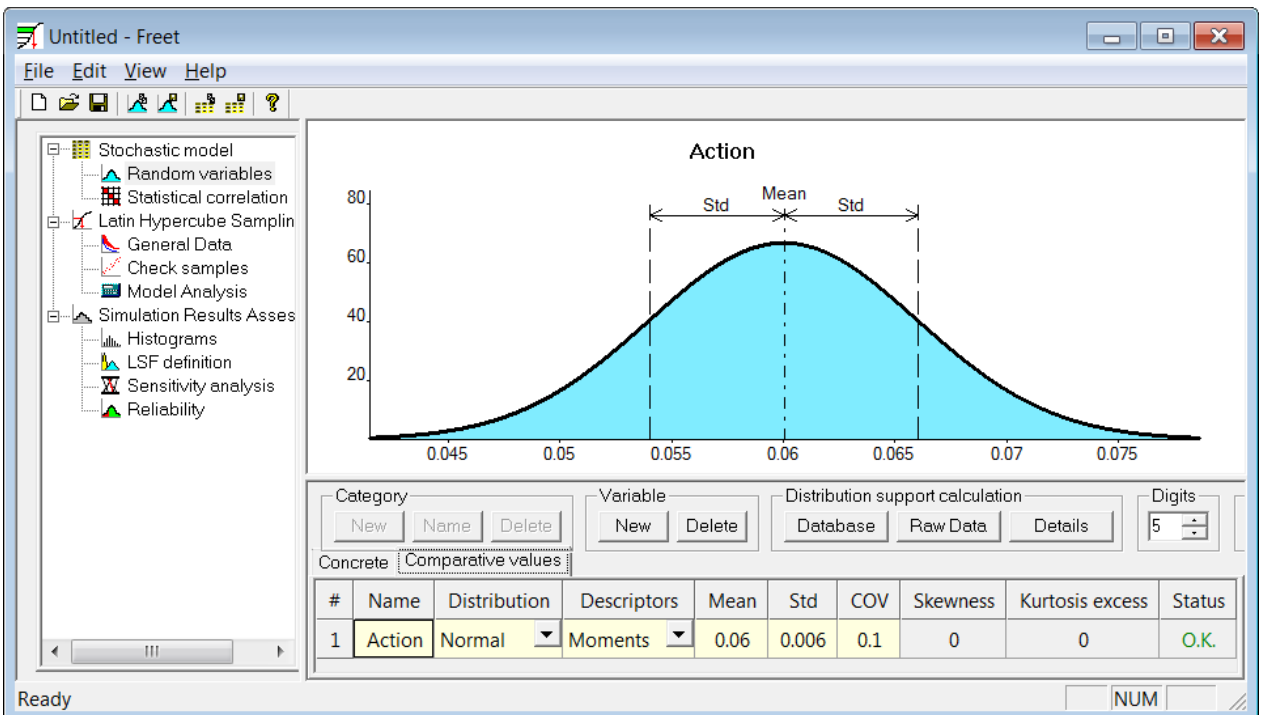
Figure 32 FREET – definition of a new comparative value named "Action"

Afterwards, it is necessary to perform sampling of this comparative value. Select **Latin Hypercube Sampling │ General Data**, check **Comparative values only** and click **Run**. Proceed to **Simulation Results Assessment │ LSF definition** and define new limit state function named reliability; see Figure 33.
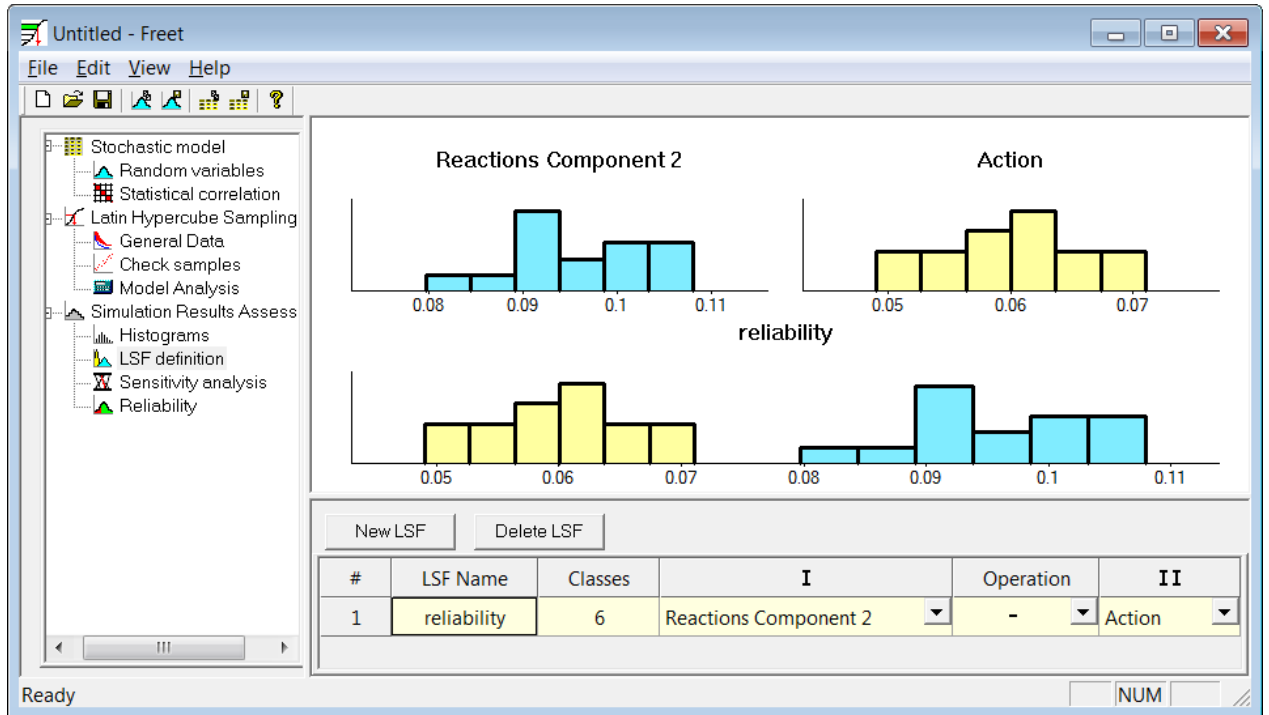
Figure 33 FREET – definition of a new limit state function

Finally, go to **Simulation Results Assessment │ Reliability** to see, that probability of failure of this structure to defined load is 0.00014886 which corresponds to reliability index β = 3.6173.
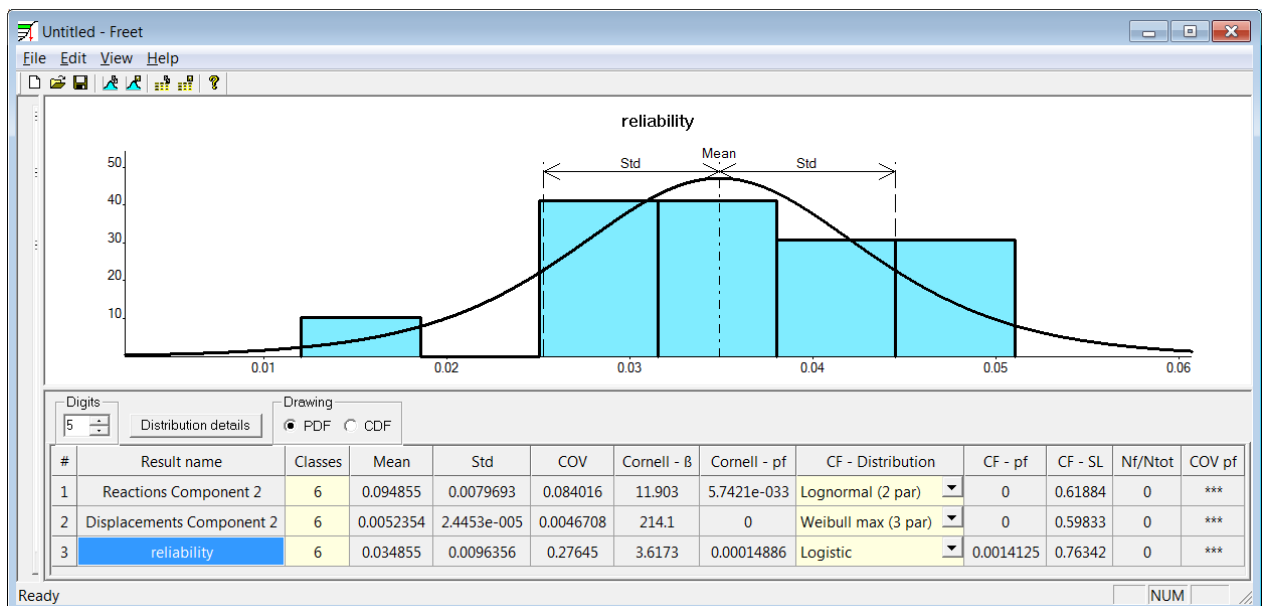
Figure 34 FREET – reliability assessment.

3 RLACS STUDIO – ADVANCED FUNCTIONS

3.1 Files in RLACS project folder

Description of files listed below is related to the tutorial example from the section 2. Red font color denotes files of particular importance.

task.cc2   ATENA binary file - exact copy of the selected basic task

task.cct   ATENA text file having specific syntax describing all necessary information to reconstruct the basic task (job name, topology, macroelements, material description, analysis steps ...). This file does not contain FE mesh.

Exactly same file can be generated when the basic task is opened in **ATENA 2D** and following **File │ Export │ CCT Format**.

To create *.cc2 file from *.cct start **ATENA 2D**, click **File │ Import │ CCT Format**.

This procedure does not work if the layers with smeared reinforcement are used

task.ccm   ATENA text file with description of the materials in the basic task

Exactly same file can be generated when the basic task is opened in **ATENA 2D** and following **File │ Export │ CCM Format**. This file cannot be imported back.

task.cco   ATENA text file with description of monitors in the basic task

Exactly same file can be generated when the basic task is opened in **ATENA 2D** and following **File │ Export │ CCO Format**. This file cannot be imported back.

task.ccp   ATENA text file with description and coordinates of the finite element mesh

Exactly same file can be generated when the basic task is opened in **ATENA 2D** and following **File │ Export | CCP Format | i.p.**

coordinates. This file cannot be imported back.

task001.cc2 ... task015.cc2

ATENA binary files. These files contain randomized input.

task001.ccm ... task015.ccm

ATENA text files. These files contain randomized values of material parameters.

task001_r.cco ... task015_r.cco

ATENA text files. Similarly to task.cco, these files contain values of monitors logged after iterations. Individual files are created when analysis of particular task is finished.

task001_r.cc2 ... task015_r.cc2

ATENA binary files. Individual files are created when analysis of particular task is finished. The difference from **task001.cc2 ... task015.cc2** is that these files contain results and their size is hence bigger.

project.prj RLACS Studio binary file - project

project.log and **task.templog** RLACS Studio text files with time and description of all operations (parsing, randomization, running tasks in **ATENA** ...)

task.fre FREET text file containing values of randomized variables, correlation matrix and monitors

task_bkp.fre, task_abkp.fre FREET backup files of **task.fre**

Msg.txt, **Out.txt** RLACS/ATENA text log files with information about analysis progress and residuals for all time steps and all randomized tasks

In.txt RLACS/ATENA text input and log file

Err.txt RLACS/ATENA text file containing errors and warnings

3.2 Program menu – explanation of program functions

3.2.1 Start

Dock

Exit Terminates program **RLACS Studio**.

3.2.2 Project

New Starts the **New project** dialog (see Figure 8).

Open Offers a list of projects located in currently selected project folder. If the project folder is empty, then warning message "Cannot find any projects" appears.

Save Saves current project.

Rename Starts a dialog, where the project name can be changed and the additional comments can be added. Project folder will be renamed automatically. Same dialog can be opened choosing **Options | Project**.

Close Closes current project and the introductory window appears (Figure 6)

Delete variants Deletes all *.cc2 files except for the basic task, and all randomized *.ccm files. Monitor files (*_r.cco) are kept; to delete them, it is necessary first to Clear results and to Delete variants afterwards – CCO měly by se taky smazat.

Clear results Deletes calculated tasks (*_r.cc2) and monitor files (*_r.cco). Monitor files are deleted only when calculated tasks are in the same folder.

Delete Deletes whole folder of the selected project. Currently project cannot be deleted.

Import

Export

Archive

Restore

3.2.3   Basic task

Import .cc2   Imports basic deterministic task. Exact copy of selected file will be copied into project folder.

Open .cc2   Opens basic task (its copy in project folder) in **ATENA 2D**. This command is useful when one wants to verify, that the correct file has been chosen or wants to modify it.

3.2.4   Randomization

Selector   Starts selector – see section 2.5. The basic task must be selected first.

Materials

   Randomization   Starts **FREET** and only those materials and their parameters, which have been previously chosen in **Selector** are offered for randomization.

        In the end, to create randomized ATENA input files select **Latin Hypercube Sampling | Model Analysis | OK**. ATENA inputs are generated automatically and the model analysis starts.

        Compare to **Randomize materials + Generate inputs**.

   Randomize materials   This function is similar to **Materials | Randomization**, except that at the end only randomized **ccm** files are created and the user is returned back to the main RLACS Studio window.

   Generate inputs   Randomized ATENA inputs based on generated **ccm** files are crated and the model analysis starts.

Input

   Randomization   This function is similar to **Materials | Randomization** (description above) except that all selected data are offered for randomization (cross-section of reinforcement bars, topology, etc…).

        In the end, to create randomized ATENA input files select **Latin Hypercube Sampling | Model Analysis | OK**. ATENA inputs are generated automatically and the model analysis begins. The FE mesh is generated automatically on the beginning of the non-linear analysis of every task.

Even though the user randomizes only materials, the FE mesh is erased automatically. In more complex problems, where the FE mesh generation takes up a long time, it is advisable to decide, what will be the subject of a probabilistic study. If only materials, then **Materials | Randomization** should be definitely used.

Randomize input   This function is similar to **Input | Randomization**, except that at the end only randomized **cct** files are created and the user is returned back to the main RLACS Studio window.

Generate inputs   Randomized ATENA inputs based on generated **cct** files are crated and the model analysis starts.

3.2.5   Analysis

Run analyses   Starts non-linear analyses. The user is asked to select, which tasks should be calculated. The default selection is the tasks from the stack on the left side, which have not been computed yet.

Import task(s)   Starts dialog where the user selects calculated tasks (*_r.cc2 files), which should be imported to **RLACS Studio**. This function is described in more detail in section 3.7.

Import monitors   Starts dialog where the user selects monitor files (*_r.cc2), which should be imported to **RLACS Studio**. This function is described in more detail in section 3.7.

Finish   This function is active when the non-linear analysis is in progress. If this function is selected, then the remaining tasks will not be started. The currently running task will not be terminated.

Exit

3.2.6   FREET

Statistic evaluation   This function should be used when all (or some) analyses have been computed. All currently available results are exported to **FREET** for statistic evaluation. This exported data set can be updated only by exiting **FREET** and starting it again.

Reliability

3.2.7   Deterministic

Run ATENA   Starts **ATENA 2D** with an empty project.

3.2.8   ATENA Interface

Starts ATENA Interface, which is a package allowing export/import of materials, coordinates, monitors and tasks.

3.2.9   Options

General   Opens dialog, where language and working directory can be set.

Project   In this dialog the user can adjust project name, task name and index length.

Histogram   Options for the histogram view. For every monitor the user can define number of intervals, the minimal and maximal value.

3.2.10   Help

Contents

Index

About RLACS Studio Shows information about the program version.

3.3 Creating *.inp file from *.cc2

As has been already mentioned, **RLACS Studio** does not work with basic project files *.cc2 or *.cc3, but with ATENA input files *.inp. A step-by-step procedure used for exporting *.cc2 files to *.inp is explained here. Similar procedure for converting *.cc3 to *.inp is explained in the next subsection (3.4).

ATENA 2D tutorial file (Leonhard's beam) is used here as an example. To create its *.inp file, follow these steps.

- In **ATENA 2D** open a copy of file:

  …\AtenaV4\Examples\ATENA Engineering\Tutorial\Beam Iso.cc2

- In the main menu go to **Options | Settings** and in the first tab **General** check **Enable input file editing before starting FE analysis** (see Figure 35).
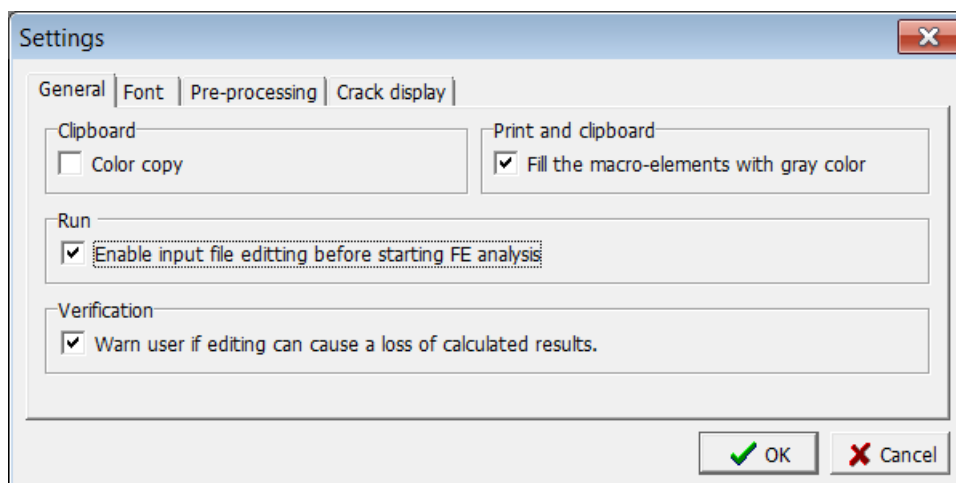

Figure 35 ATENA 2D Options | Settings

- Start FE analysis (click **Calculation | Analysis**) and before clicking on button **Analyse** make sure that the checkbox **Edit ATENA analysis input file** is checked.

- Figure 36 shows the ATENA input file for the first analysis step which appears right after button **Analyse** is pressed. Click **Select all** (or Ctrl + a) and icon **Copy** (or Ctrl + c). Use your favorite text editor to create a new text file named **Beam_iso.inp** and paste there content of the clipboard.

- Now, it is necessary to create a *.inp file for all analysis steps, not just for the first one. If you close ATENA analysis input file, the computations proceeds and as soon as the first analysis step is finished, *.inp file for the second analysis step opens (Figure 37). Copy and paste its content at the end of Beam_iso.inp. You can follow this procedure for all analysis steps or just copy and paste *.inp file of the second analysis step and change the number of the analysis step manually (i.e. change 2 for 3 … 50). This can be done only as long as all analysis steps are equal, i.e. same solution method and same loading cases with same coefficients are used).

- If the numbers of analysis steps are changed manually, it should be done at four places, but must be only at one:
  - LOAD STEP NO.**5** (header LOAD STEP NO. # optional)
  - STEP id **5** STATIC name "Load step No.**5**" (**id # compulsory**, Load step No. # optional)
  - END OF LOAD STEP NO.5 (footer END OF LOAD STEP NO. # optional)

- To avoid opening of unwanted *.inp file at the beginning of each step, uncheck **Edit ATENA analysis input file**.

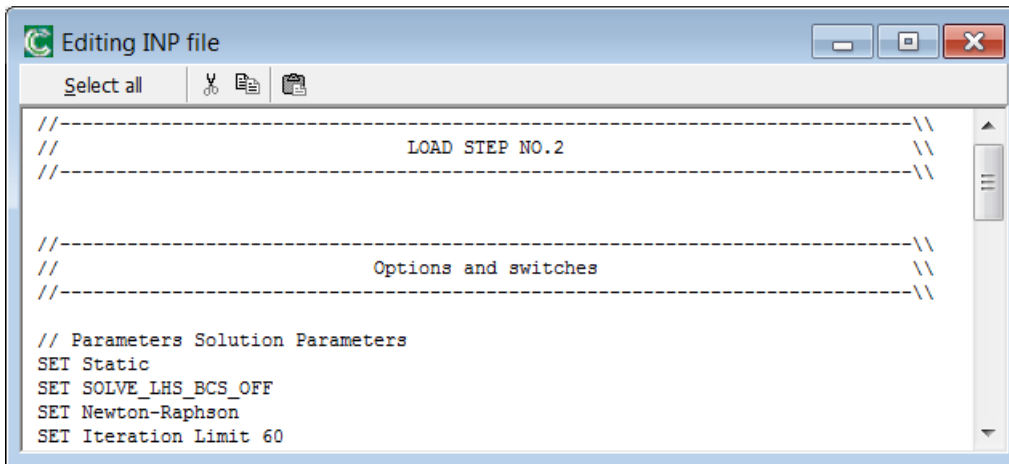**Figure 36 ATENA 2D *.inp file for the first analysis step**

Figure 37 ATENA 2D *.inp file for the second analysis step

3.4 Creating *.inp file from *.cc3

How to create an ATENA input file from ATENA 2D file *.cc2 has been explained in the previous section, 3.3. In this subsection a similar procedure for converting ATENA Engineering 3D files *.cc3 to ATENA input files *.inp is to be explained.

ATENA Engineering 3D tutorial file (Leonhard's beam) is used here as an example. To create its *.inp file, follow these steps.

- In **ATENA Engineering 3D** open a copy of file:

  …\CervenkaConsulting\AtenaV4\Examples\ATENA Engineering\Tutorial\ \Shear beam 3D.cc3

- Click **Data | Save INP files …** and following dialog appears. If you selected **Individual**, an individual input file would be created for every analysis step. Therefore choose option **Together** to create just one input file.

- Select target path and name for exporting and confirm.

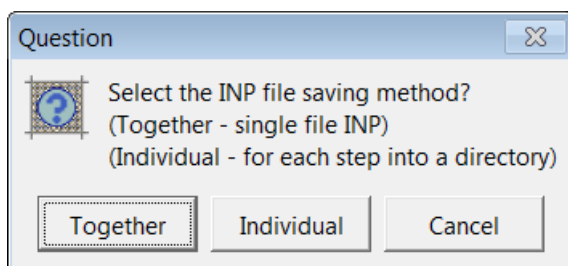Figure 38 ATENA Engineering 3D export dialog

3.5 Selection of random variables in the *.inp file

As has been shown in the tutorial example if a virgin ATENA input file *.inp is used, neither materials nor material parameters are handled to Selector. The input file contains a vast number of numerical values and therefore the user must decide and select, which values are intended for randomization.

More parameters and values can be selected in the input file and the final choice can be kept for the Selector. If more parameters are to be selected, it is advisable to create several groups, which will improve lucidity and organization. These groups will afterwards appear in the tree menu in **Selector**.

Values which should appear in Selector and should be randomized should be marked by special string, which can be easily distinguished by a program parser. If a material parameter or a value should be transferred to Selector and randomized, put following string just before the numerical value:

```
/*# any comment describing variable */
```

To gather e.g. material parameters into a group, put following string before the first one.

```
/*## any comment describing group */
```

All values marked `/*# comment */` which follow that string will be sorted into that group. Use multiple hashes to create a hierarchy.

```
/*### Materials */
/*## Concrete */
/*# E */
/*# fc */
/*## Steel */
/*# E */
/*### Loading */
…
```

Several examples follow.

Selection of the Young modulus:

original line: `E   3.172E+04`

modified line: `E   /*# Young's modulus */ 3.172E+04`

meaning: value `3.172E+04` named `Young's modulus` will be handled to Selector

Selection of yield stress and strain and incorporating them into group "Rebars":

original section:
```
FUNCTION
id 1
name "Function for Material Reinforcement"
type "CCMultiLinearFunction"
xvalues  0.000E+00  2.800E-03
yvalues  0.000E+00  5.600E+02
```

modified:
```
/*## Rebars */
FUNCTION
id 1
name "Function for Material Reinforcement"
type "CCMultiLinearFunction"
```

```
            xvalues  0.000E+00 /*# yield strain */ 2.800E-03
            yvalues  0.000E+00  /*# yield stress */ 5.600E+02
```

meaning: values `2.800E-03` named `yield strain` and `5.600E+02` named `yield stress` will be handled to Selector. These values will be incorporated into group named `Rebars.`

3.6 Export of monitor data to Excel

To export monitor data to **Excel**, go to the **Line chart view** and then right-click anywhere in the chart; then select **Export...**. Then the user is asked to specify path and the file name. So far, only the *.csv (colon separated values) format has been implemented.
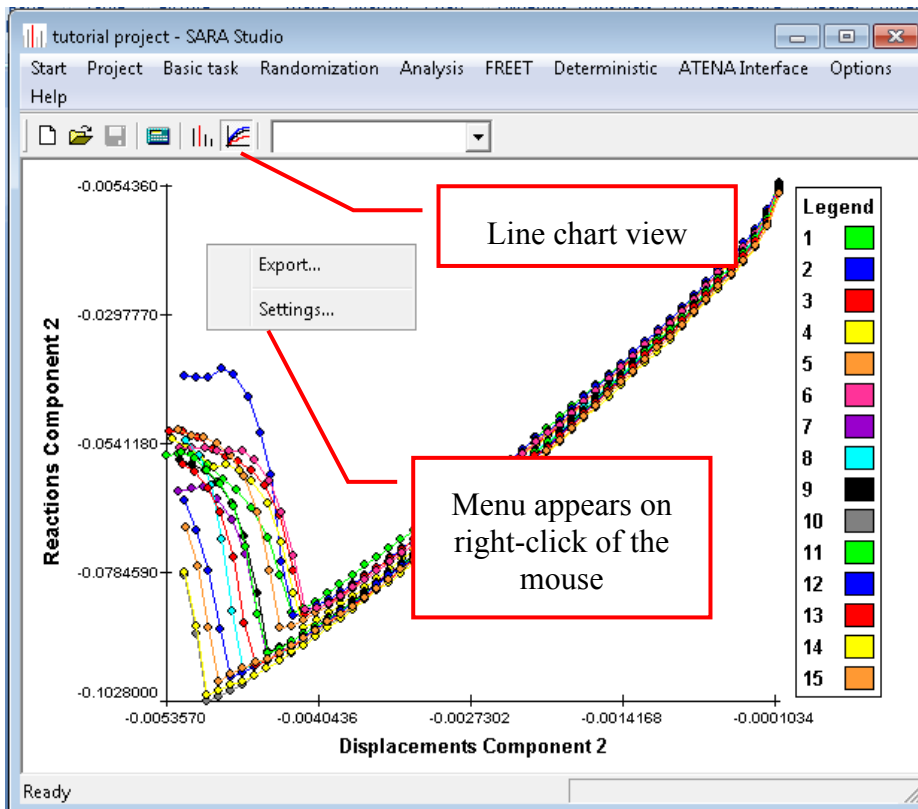

Figure 39 Export of monitor data

Figure 40 shows sample exported data. The first two lines specify the project name and path. The fourth and fifth rows carry labels of the axes of the plot (i.e., the monitor tag or the step number).

The remaining cells contain data that are currently potted in the chart. Every computed task has its job number and sets of X and Y values sorted in rows.

Sometimes, it is better to have the data in columns, instead of rows. To easily switch rows and columns (i.e. to transpose the data), select and copy the range of data, which should be converted. Then right-click into the cell, where the cell of the first row and column should be inserted, and select **Paste Special** and check option **Transpose**.
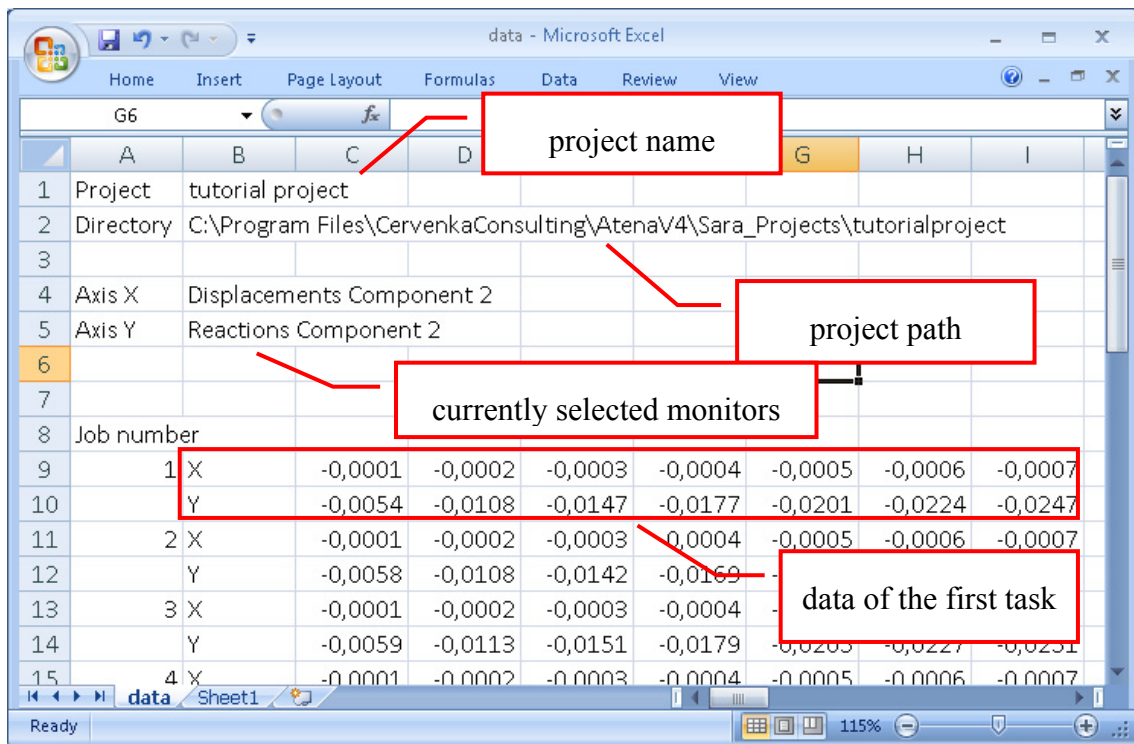
Figure 40 Exported data in Excel

3.7 Running analyses outside RLACS and importing results

Analysis of individual randomized task can be driven either from **RLACS Studio** or individual tasks can be calculated separately and then imported back to RLACS. This can be useful especially if:

- the computer has more processors/cores (significant speed-up of probabilistic analysis)

- there is likelihood, that the analysis stops due to violation of convergence criteria (next task does not start unless the previous one sends signal to **RLACS**, that the analysis has finished, which happens when warning window is closed)

- analysis setup of some tasks has been changed (e.g. refined or added analysis steps) and the task has to be computed from scratch or finished

To run analyses independently of **RLACS Studio** proceed following steps.

1) create an exact copy of the randomized task and rename it by adding "**_r**" to the file name (e.g. if the original file is named problem001.cc2, then the copy should be problem001_r.cc2)

2) perform analysis in **ATENA 2D** ("save data after each analysis step" should be checked)

3) either export results (monitors) or import finished task from **RLACS** (to be explained afterwards)

In all other cases, the ATENA 2D file with file name followed by "**_r**" should already exist. Then make intended subtle changes (adding or refining analysis steps, changing tolerance criteria ...) and proceed with points 2 and 3 from above.

There are two different ways to import calculated results to **RLACS Studio**.

- The first one uses the *.cco file, which contains only monitor data. This file is far smaller than the ATENA *_r.cc2 file. To create it open *_r.cc2 file in ATENA and click **File | Export | CCO Format**. To import results to RLACS Studio, open the project in RLACS Studio, then follow **Analysis | Import monitors ...** and the dialog window asking for specification of *.cco file appears. Choose appropriate file(s) and click **OK**. It is possible to import more files at once. Make sure that you import files from the correct directory.

- The second approach uses import of *_r.cc2 files. Open your project in **RLACS Studio**, then in the program bar select **Analysis | Import task(s)...** and a dialog appears. Select all calculated tasks (*_r.cc2 extension) that you want to import and click **OK**. This operation might take a few minutes, depending on the number of imported tasks and on their size, because what happens is that **RLACS Studio** opens each individual task, exports monitors into the *_r.cco file and closes that task. If many tasks are selected or their size is big (50 MB or more), **RLACS Studio** might stop responding, so in that case it is advisable to import just few tasks and repeat that procedure again.

3.8 Topology randomization – geometry

In order to randomize geometry (position of joints) of the specimen, it is necessary to select (check) particular joints in selector. The numbers of particular joints can be determined according to their position, or one can open the basic task via RLACS interface. To achieve that, click **Basic task | Open basic .cc2**.
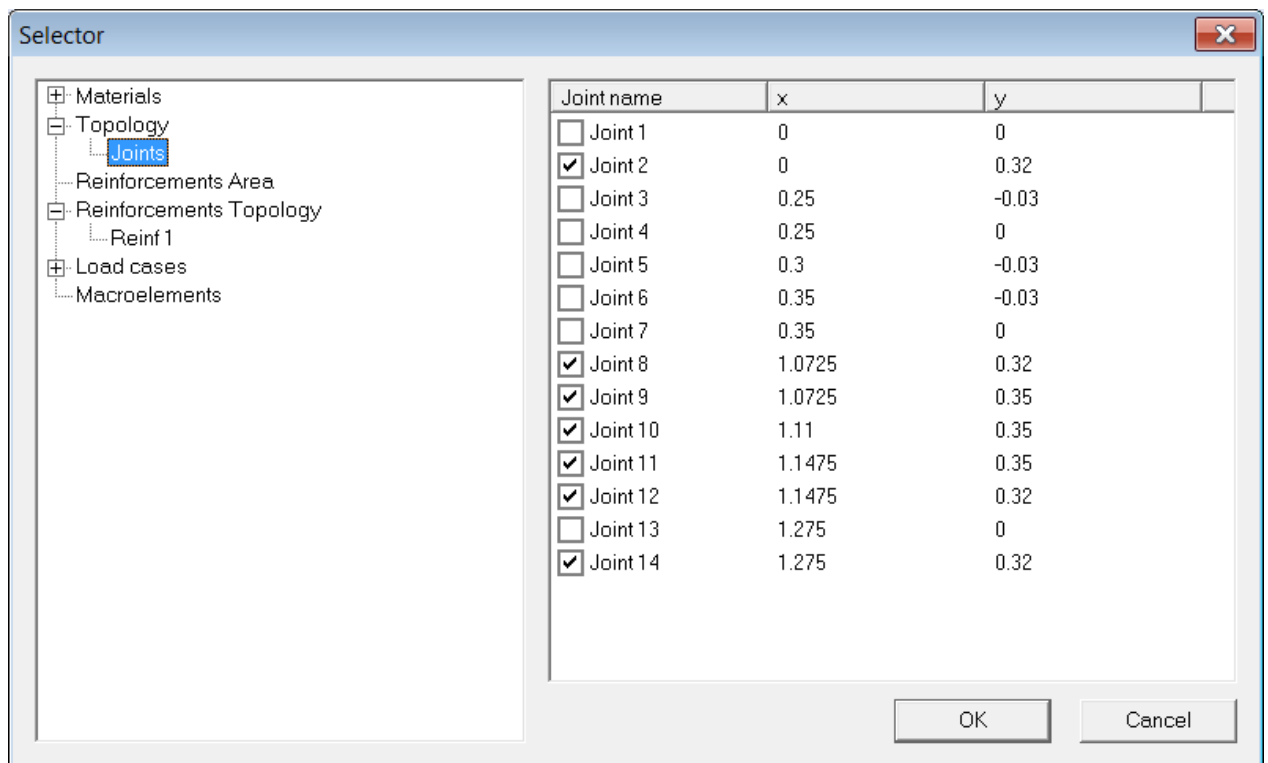

Figure 41 Selector – list of joints in the basic task.

In the example shown in Figure 41, only joints located at the top surface of the beam and joints of the top loading plate are selected. This example can demonstrate sensitivity of obtained results to changed height of the specimen. In the next step, only shift in the vertical direction is allowed. Next, it is necessary to achieve that the vertical shift of all randomized joints is the same. This can be done via correlation matrix, see Figure 43. This matrix is full of ones, which means that displacement of particular joint influences all other selected joints to move in the same direction with the same magnitude. Joints 9, 10 and 11 were selected in order to keep constant thickness of the loading plate. Finally it is necessary to check the samples to verify, that all monitoring points are properly located. In this particular example this means, that the monitoring point measuring the external force is close to node number 10 and that this monitor is inside the structure. As expected, the sensitivity among vertical shift and maximal reaction is positive and almost linear (see Figure 46).
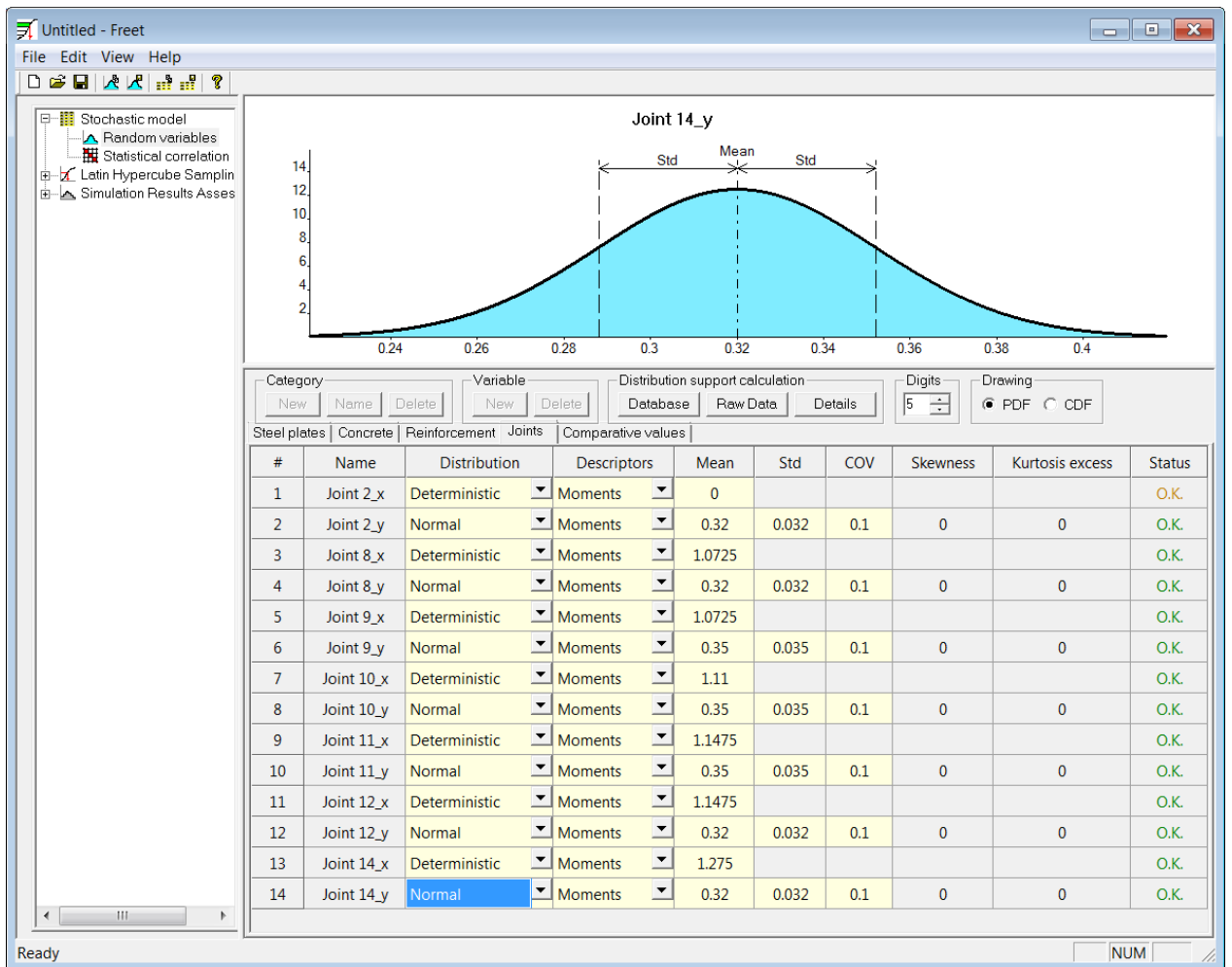

Figure 42 Freet – randomization of joints' coordinates.
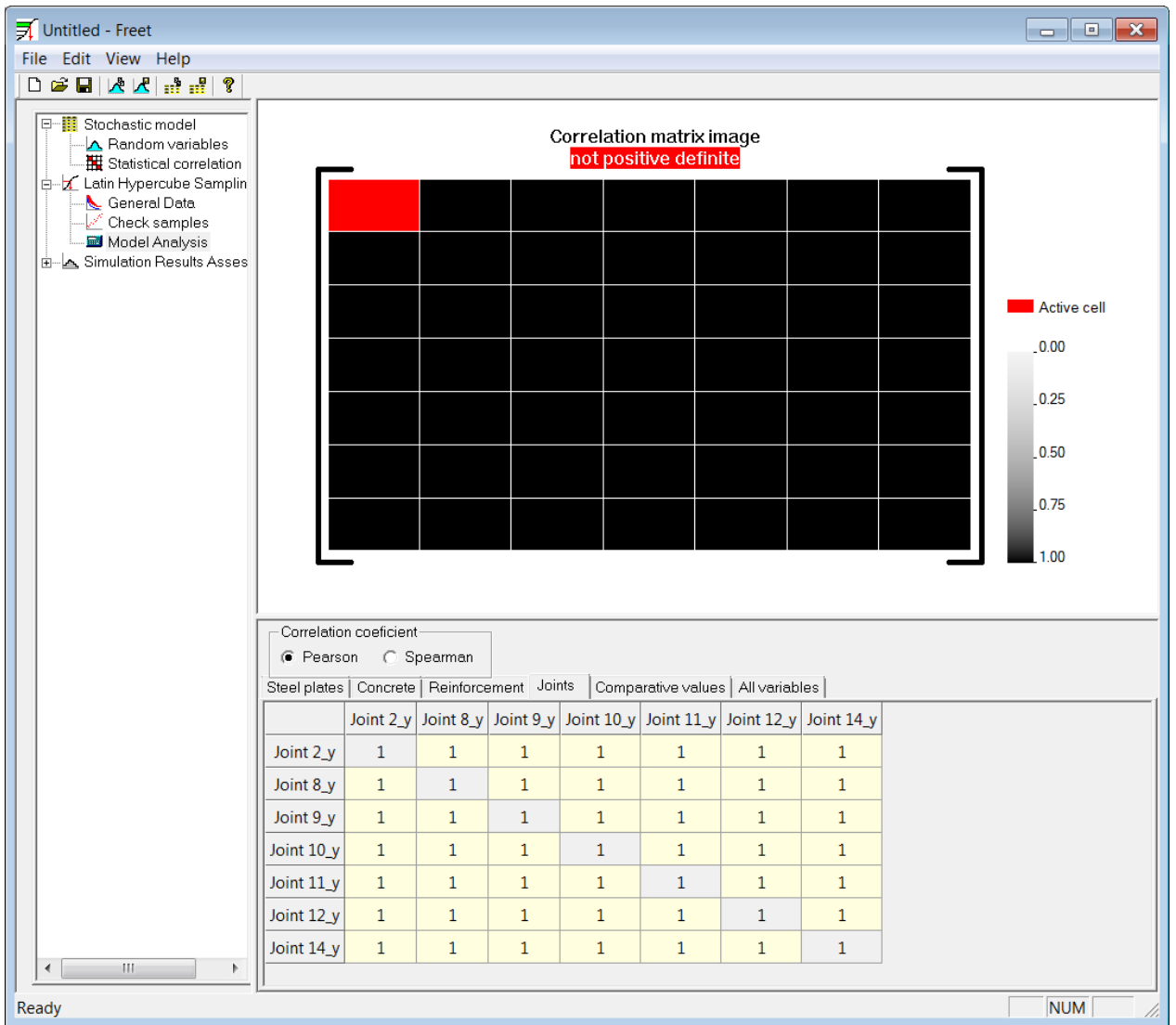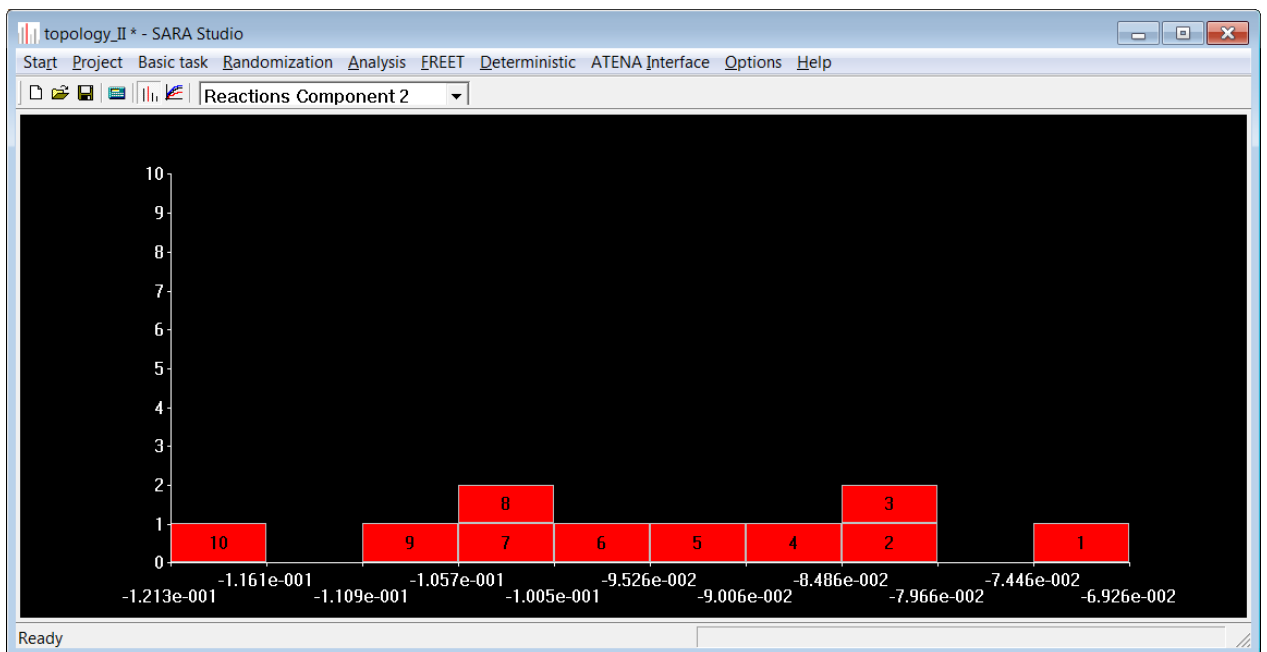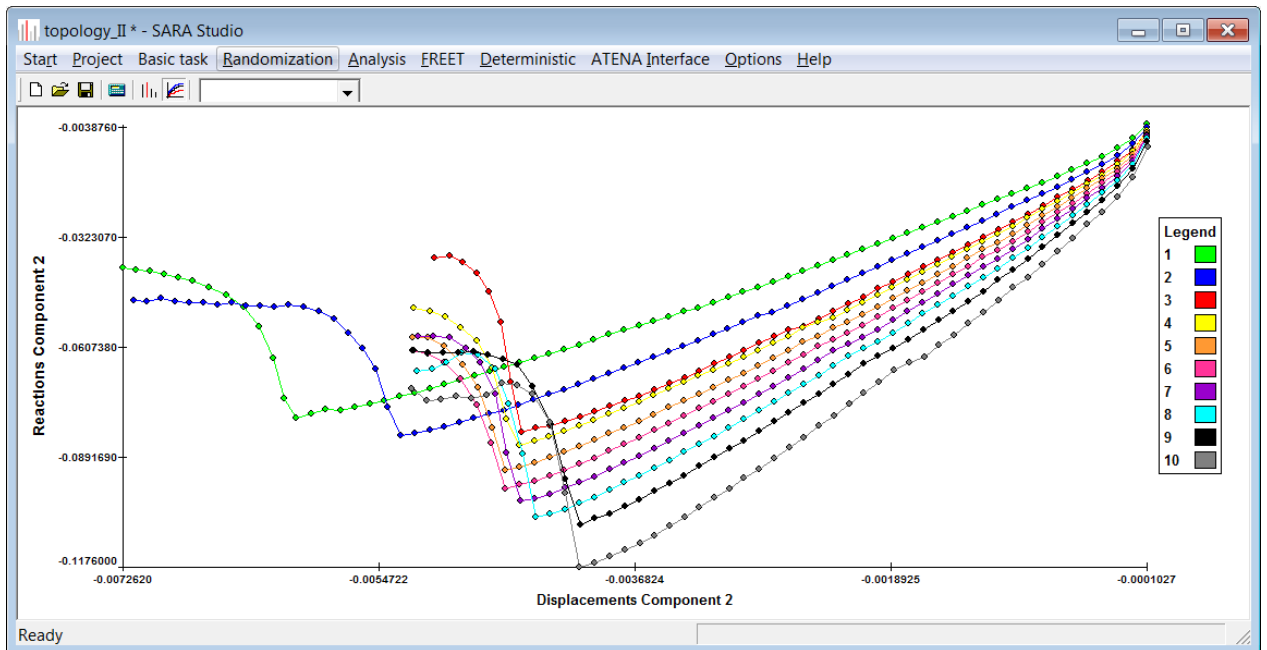
Figure 43 Freet – correlation matrix.

Figure 44 Results according to maximal value of reaction

Figure 45 Obtained LD diagrams
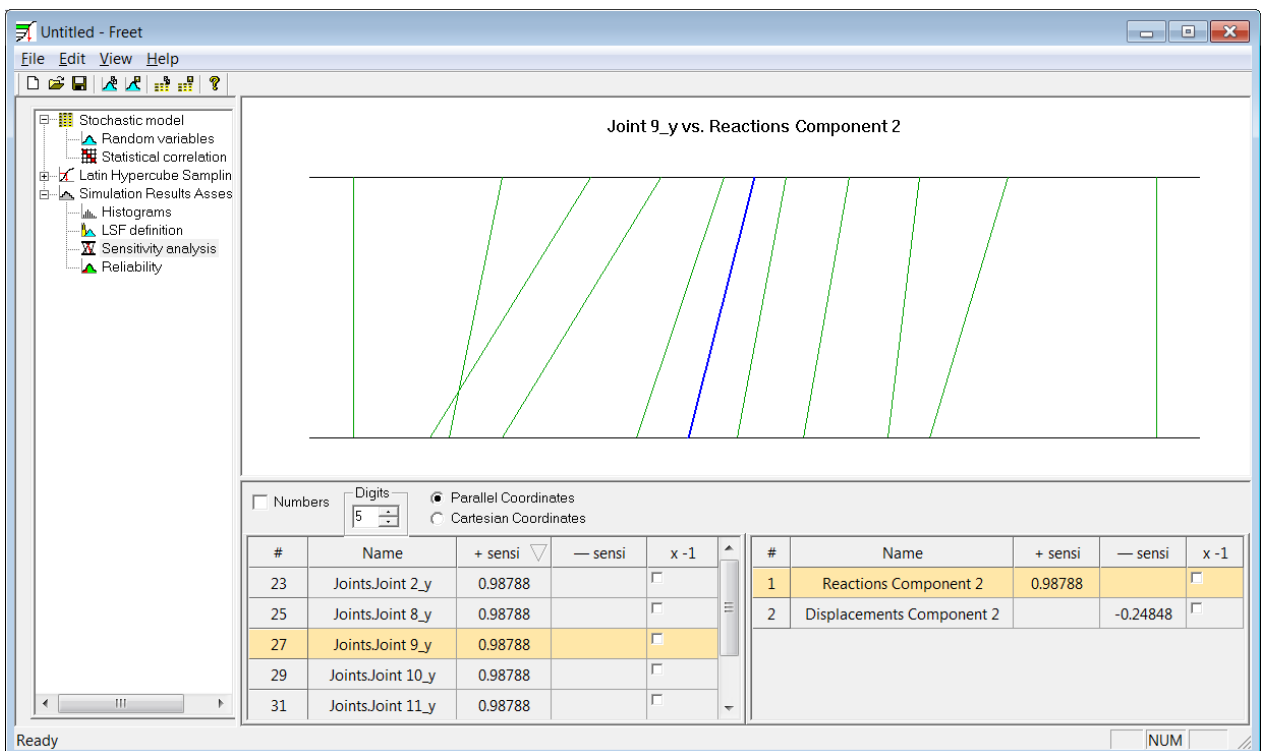
Figure 46 Sensitivity of the problem

List of problems which might appear as a result of topology randomization

- wrong position of monitoring points (either the monitoring point is outside of specimen or the wrong integration point or node is selected)
- singularity of the stiffness matrix caused by sticking out reinforcing bars

- insufficient mesh density

- problems in analysis setup or analysis steps (solution method, length of analysis steps, number of analysis steps)

3.9 Topology randomization – reinforcing bars

3.9.1 Reinforcement area

It is possible to randomize the cross-sectional area of all reinforcing bars of the basic task. First, select bars you intend to randomize in selector (Figure 47 shows only one item) and then confirm your selection by clicking **OK**. Next, click **Randomize inputs …** and **Freet** starts. In the **Reinforcements Area** tab define PDFs, then generate inputs and finally perform probabilistic analysis.
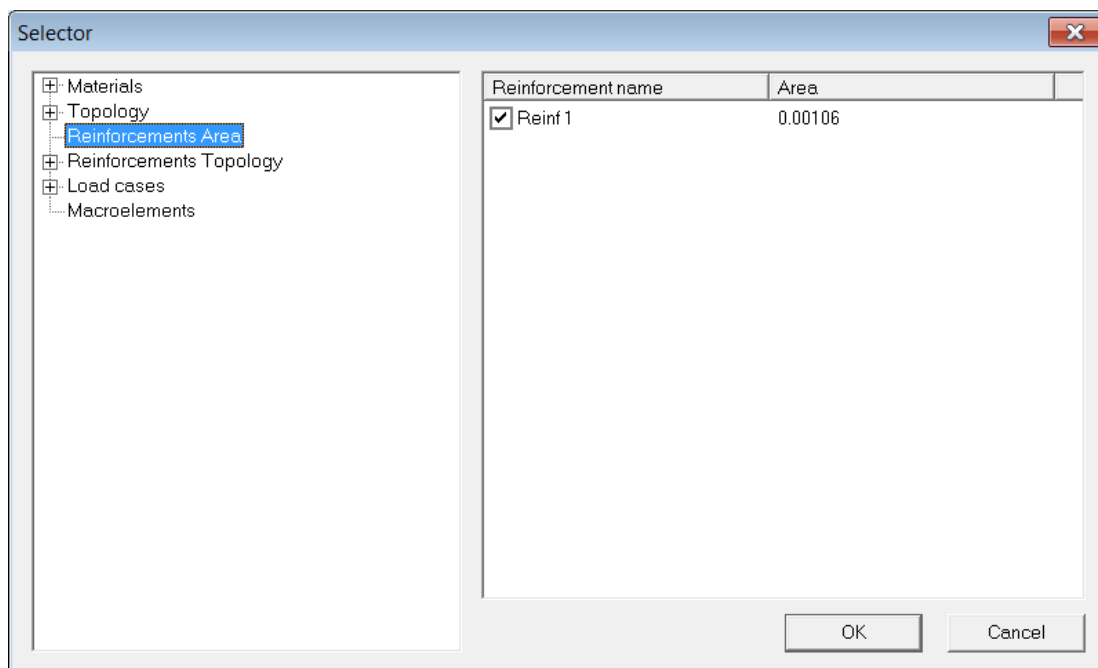

Figure 47 Selector – Reinforcement Area randomization

3.9.2 Reinforcement topology

The position of reinforcing bars can be randomized in two different ways. First option (Section 3.9.2.1) enables movement of the selected reinforcing bar as a rigid body in both horizontal and vertical directions (no rotation or stretching is allowed); the second option (Section 0) gives the user more possibilities.

3.9.2.1 Bar shifting

This section describes the first possibility of randomization of the position of reinforcing bars. In **Selector**, click on **Reinforcement Topology** in the tree menu on the left side of the window and the list of all reinforcing bars appear on the right side of same the window. Select (check) all reinforcing bars, which position should be randomized. Randomization type should be kept default = **Bar shifting**, see Figure 48.

Next, the user can either proceed to **Freet** (click **OK** to terminate Selector and then click **Randomize inputs …**) or the user can choose whether both components of movement (horizontal and vertical shift) should be randomized. Figure 49 shows that only vertical component was selected.

In **Freet**, tab appears for every selected reinforcing bar and only selected components are offered.
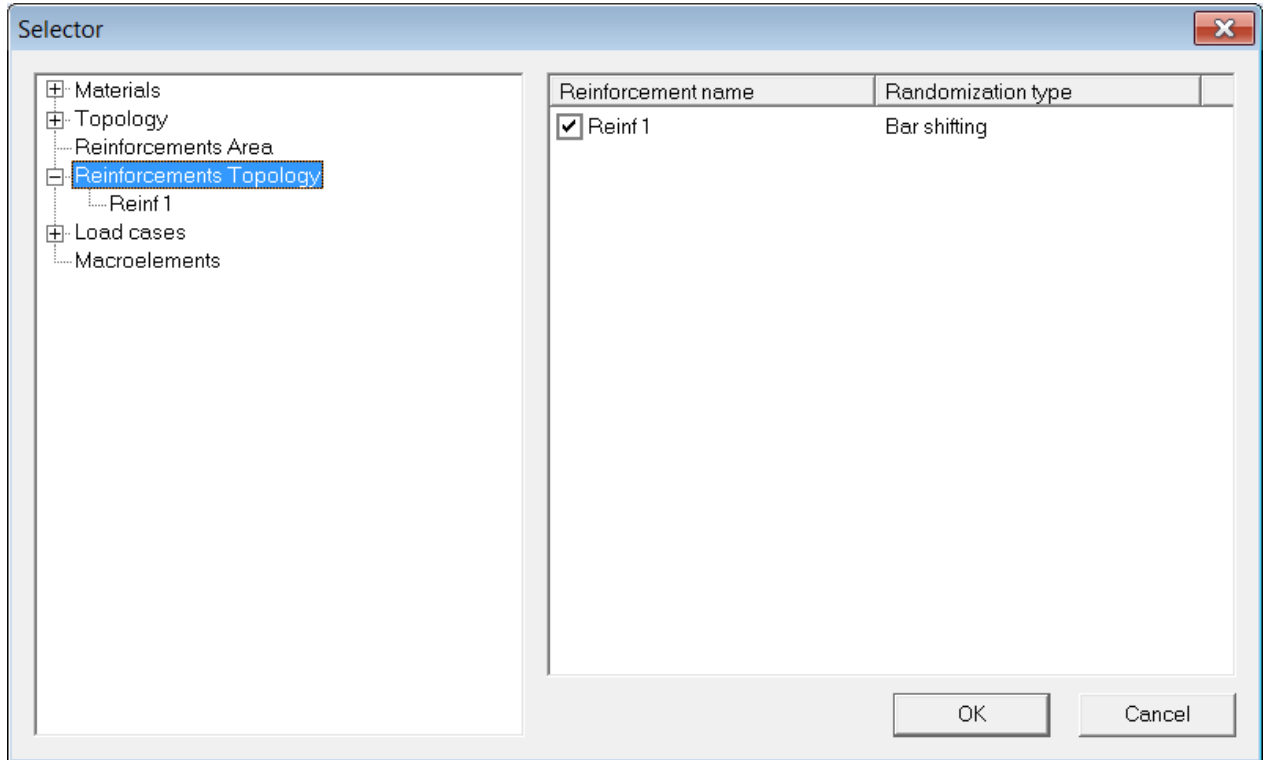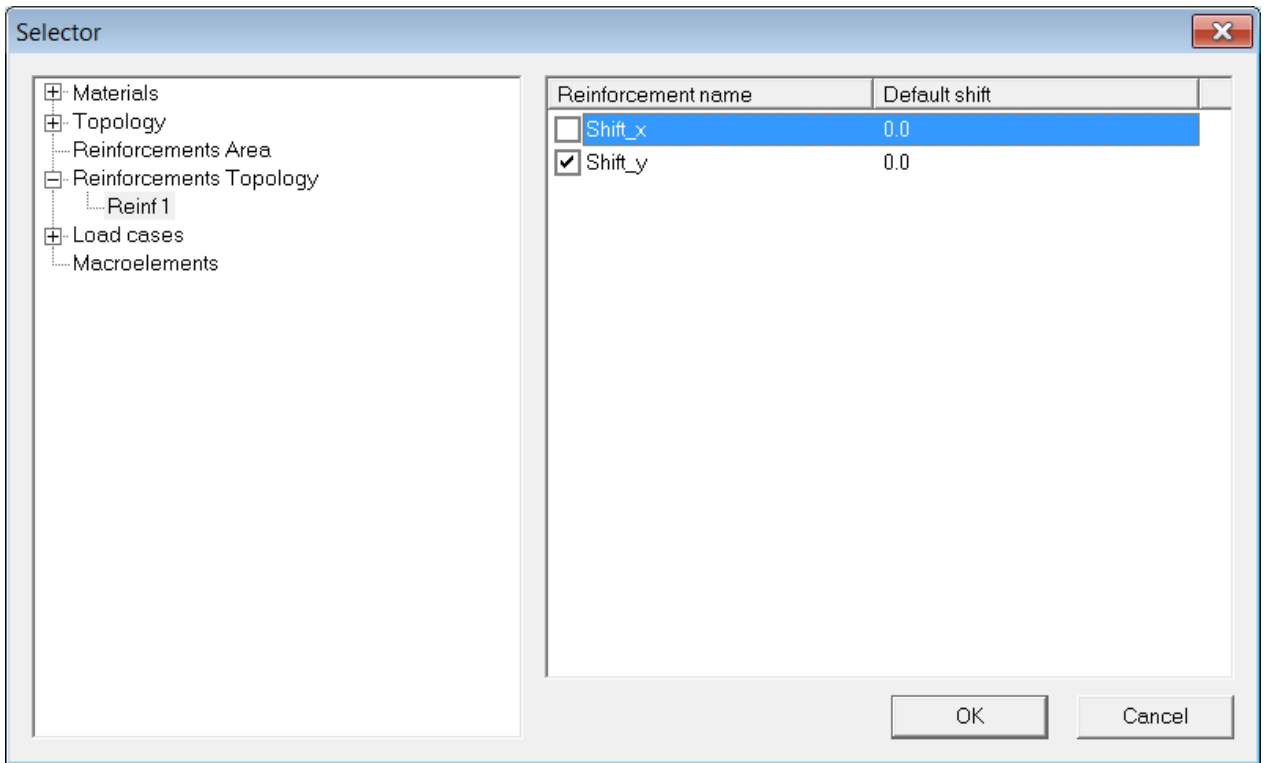

Figure 48 Selector – Reinforcement Topology

Figure 49 Selector – Reinforcement Topology – Reinf1 is expanded to display vertical and horizontal component of shift.
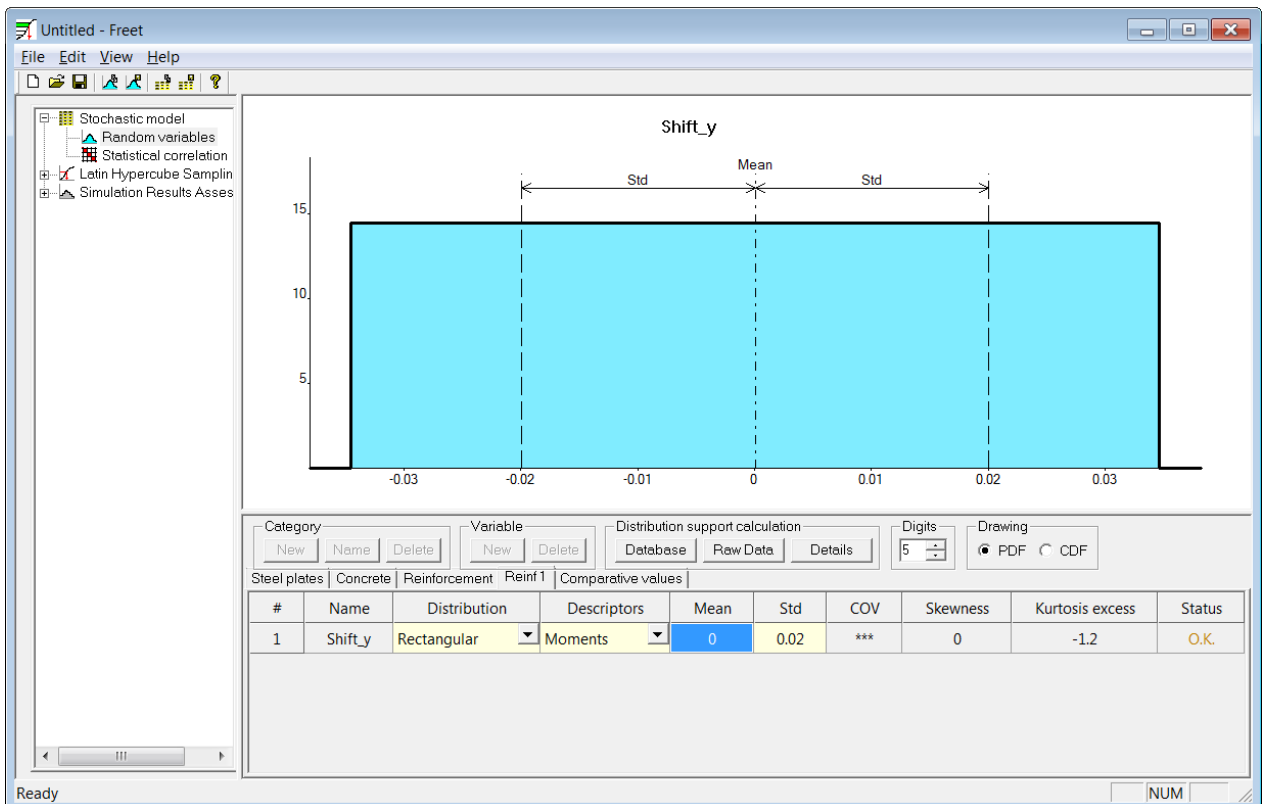

Figure 50 Freet – randomization of rebar position

Before the analysis, it is advisable to check, that all reinforcing bars are inside of the structure.

3.9.2.2 Free move

The second possibility is to randomize position of all points defining selected reinforcing bars. First, similarly to 3.9.2.1, it is necessary to select all reinforcing bars, which position will be randomized. Click on **Bar shifting** to change the randomization type to **Free move**, see Figure 51.

Then expand **Reinforcement Topology** in the tree menu and select reinforcing bar, which position is to be randomized. On the right side of the window appears the list of points with their coordinates, which define topology of that rebar. Again, check those points, which should be later offered in **Freet**, see Figure 52.
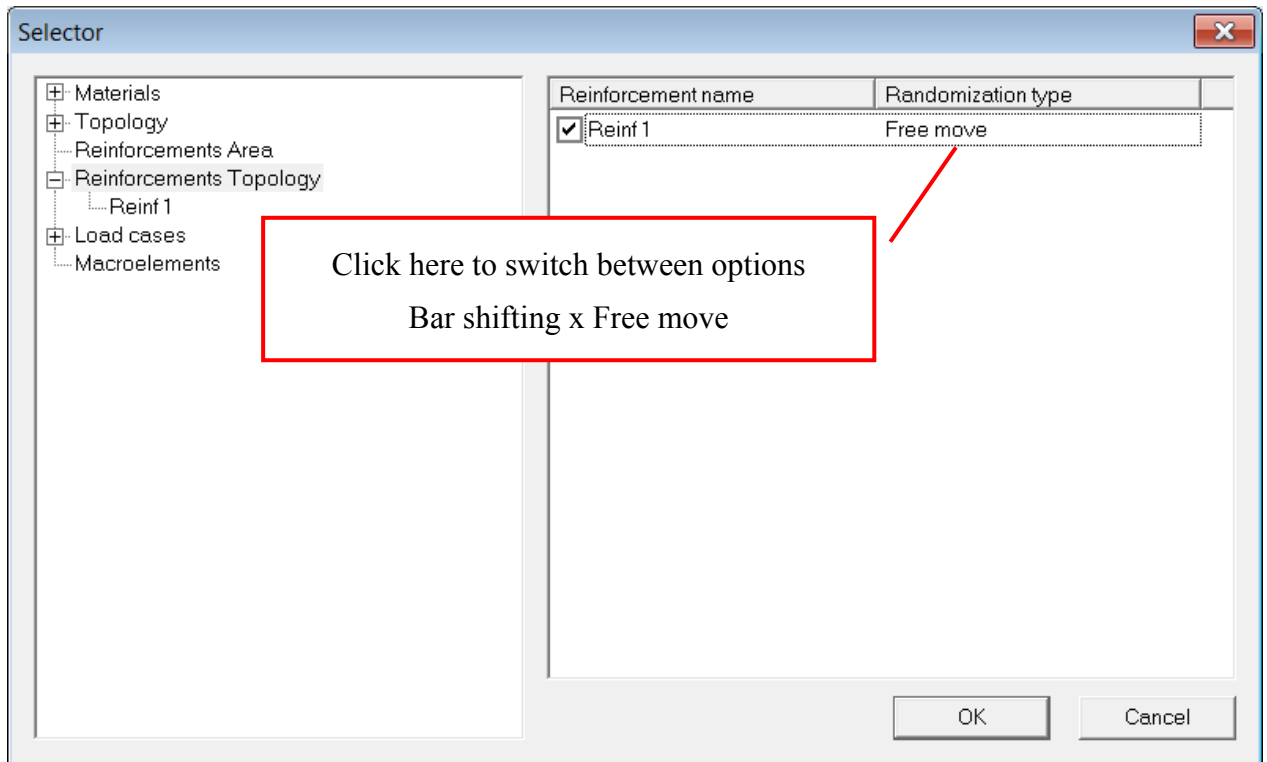

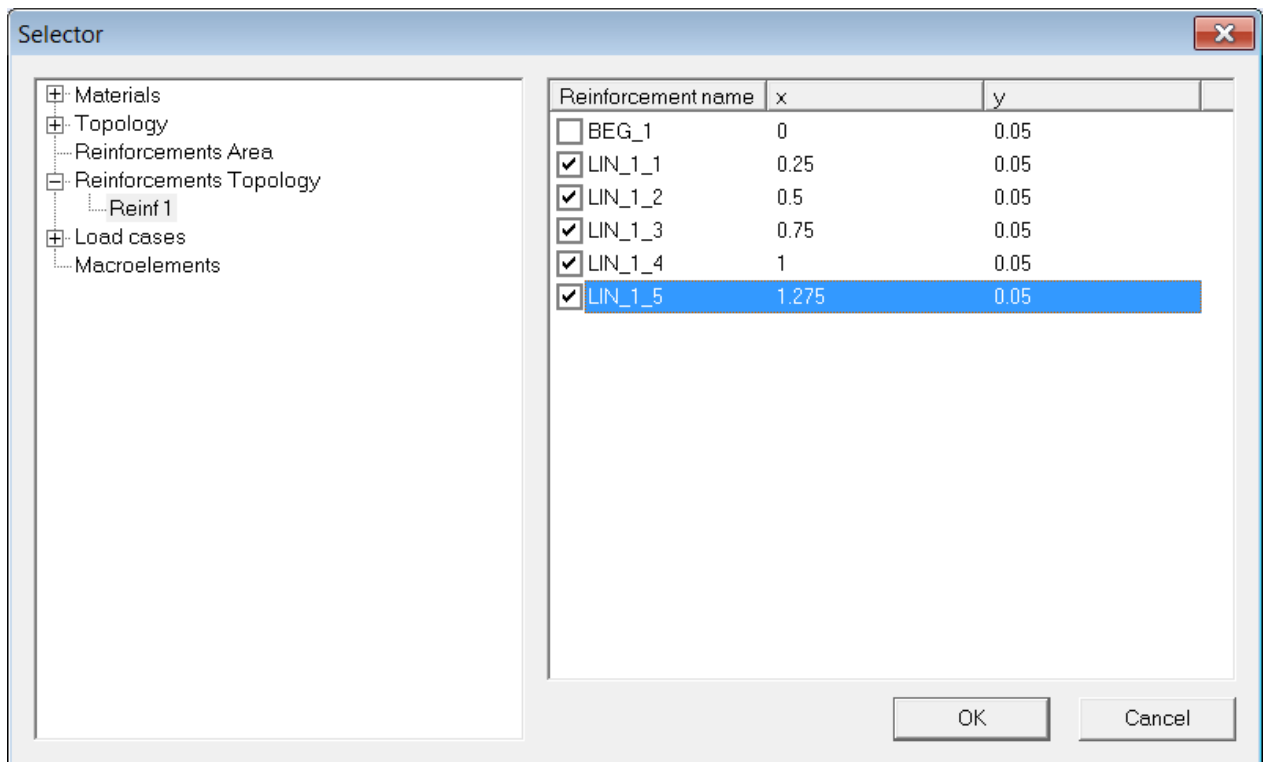**Figure 51 Selector – Reinforcement Topology**

Figure 52 Selector – Reinforcement Topology – Reinf1 is expanded to display all points defining reinforcing bar

3.10 Correlation among variables

http://en.wikipedia.org/wiki/Correlation_and_dependence:

The difference between correlation & dependence is that a strong correlation only indicates that two (or more) data sets move together but (unlike dependence) does not establish any causal relationship. For example, IQ of a child & size of his shoe will show a strong positive correlation but this does not mean that children with larger feet have greater IQ. Both these variables however 'depend' on a third factor i.e. age of the child.

<u>**Remark – correlation among variables of opposite sign**</u>:

When filling the correlation matrix, one has to be extremely careful when the correlation among two variables of opposite sign is to be specified. The simplest demonstration of this problematic is the correlation between Young's modulus E and the compressive strength f_c. For concrete, these two characteristics are not independent, for example gives the relation between the Young modulus (in GPa) at the of 28 days and the mean compressive strength (in MPa) in the form

$$E_{c28} = 21.5 \left(\frac{f_{cm}}{10} \right)^{1/3}$$

This means that the increase of compressive strength leads to cubic increase of the Young modulus. Hence, the correlation must be positive.

But in **ATENA**, the compressive strength is a negative sign; therefore the coefficient must be negative. The positive number would have led to decreasing strength with increasing modulus.

Another example, indeed, very similar to the previously mentioned one, can be created in order to demonstrate correlation among variables, which are not shown in the same tab in **FReET**.

Again, consider a four-point bending test, where the height of the specimen should be randomized, but the distance between the lower surface of the beam and the reinforcing bar should be kept uniform. Unlike the previous example, position of joints at the top surface is deterministic, while at the bottom of the specimen their position is randomized.

The key to successful application of this type of randomization lies in using tab **All variables** in **FReET** (for reference see Figure 58). In that tab are listed all variables, which were previously selected in selector and afterwards their PDF was defined (see Figure 54 to Figure 57).
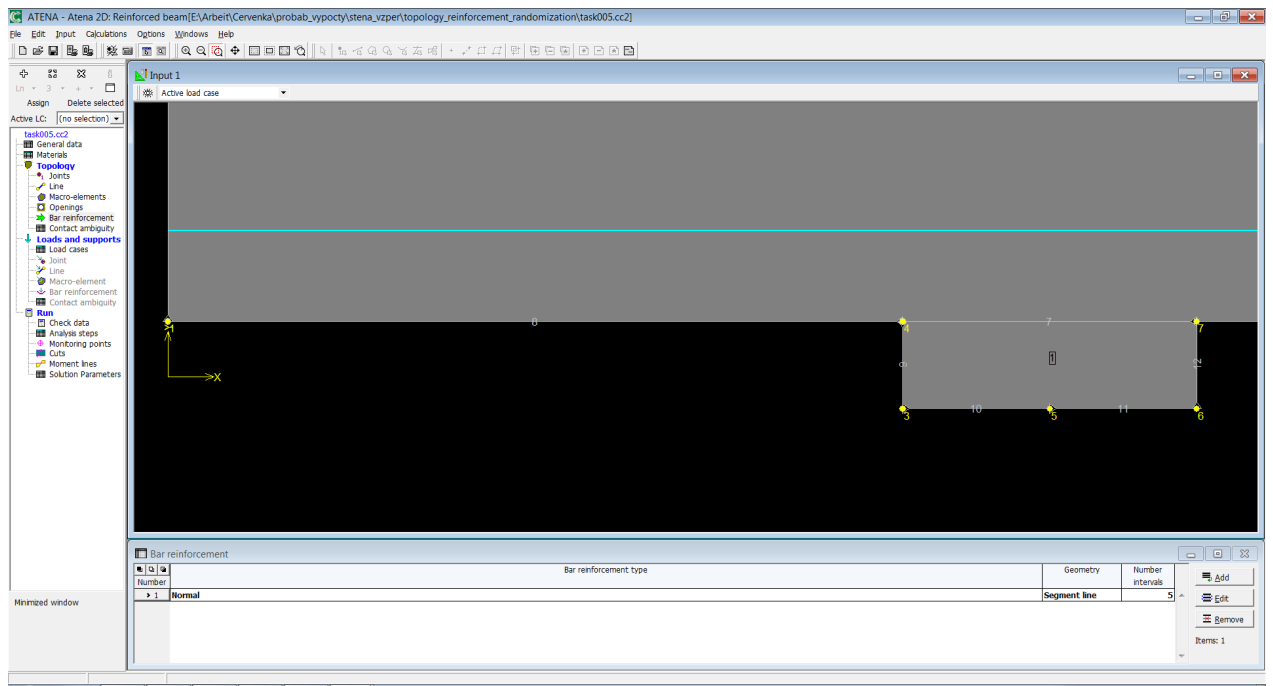
Figure 53 Obrázek ukazuje nefunkčnost znáhodnění – generovaná varianta #5 – výztuž se nehla, zatímco ostatní uzly se posunuly.

Figure 54 Selector – selection of joints located at the bottom of the beam (+ bottom loading plate)

Figure 55 Selector – vertical movement of reinforcing bars is chosen for randomization

Figure 56 FReET – vertical coordinate of all selected joints is randomized

Figure 57 FReET – vertical coordinate of reinforcing bar is randomized

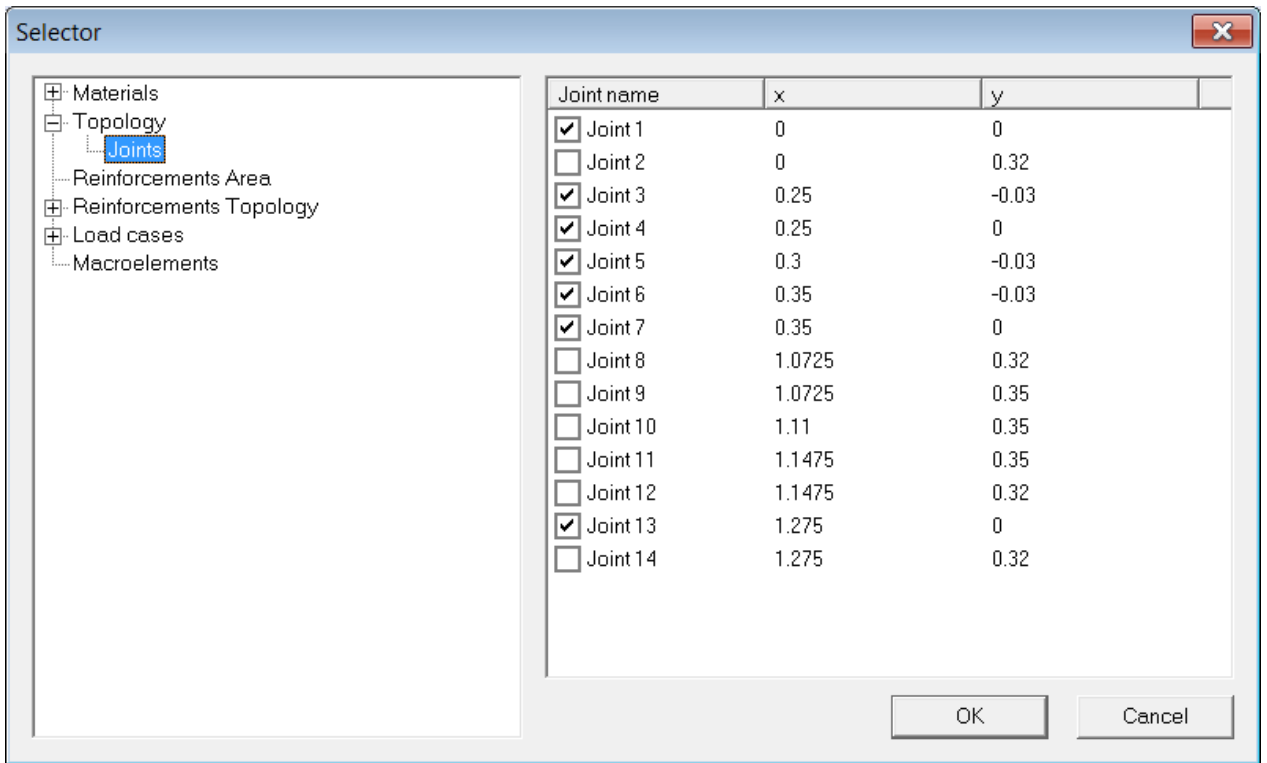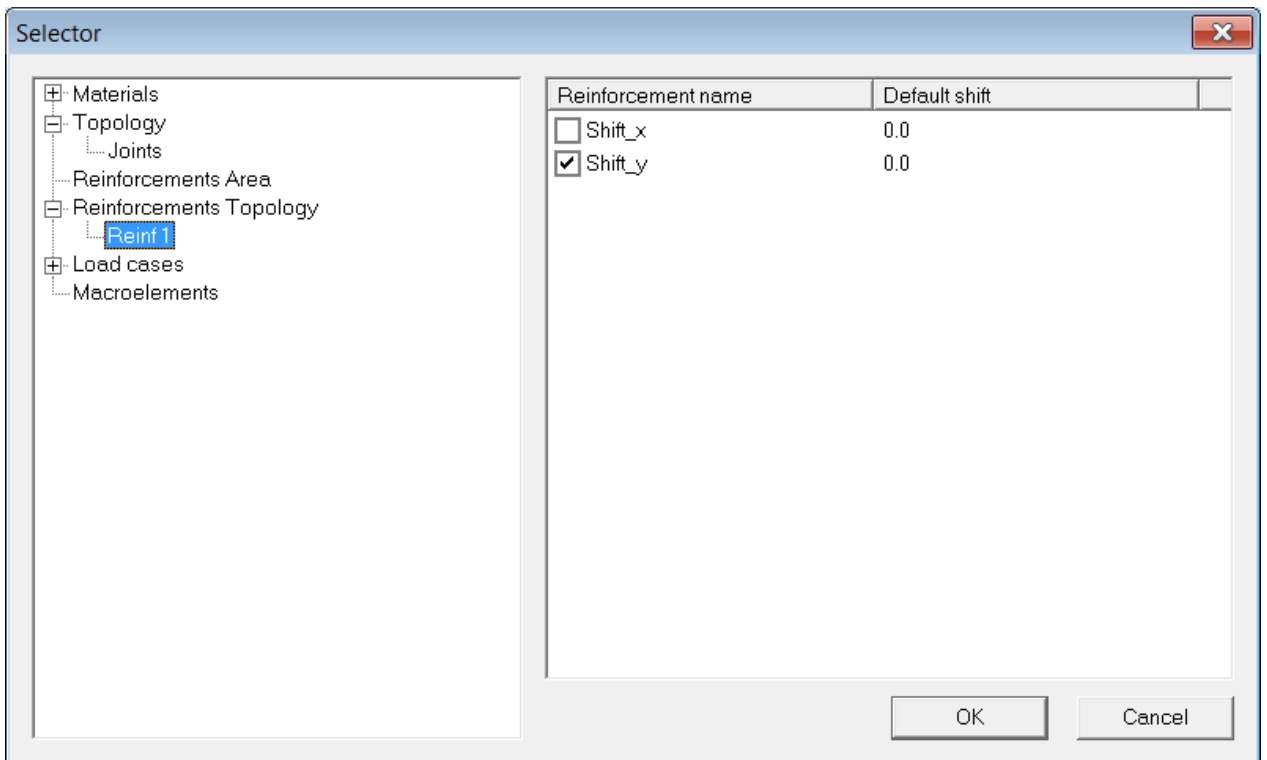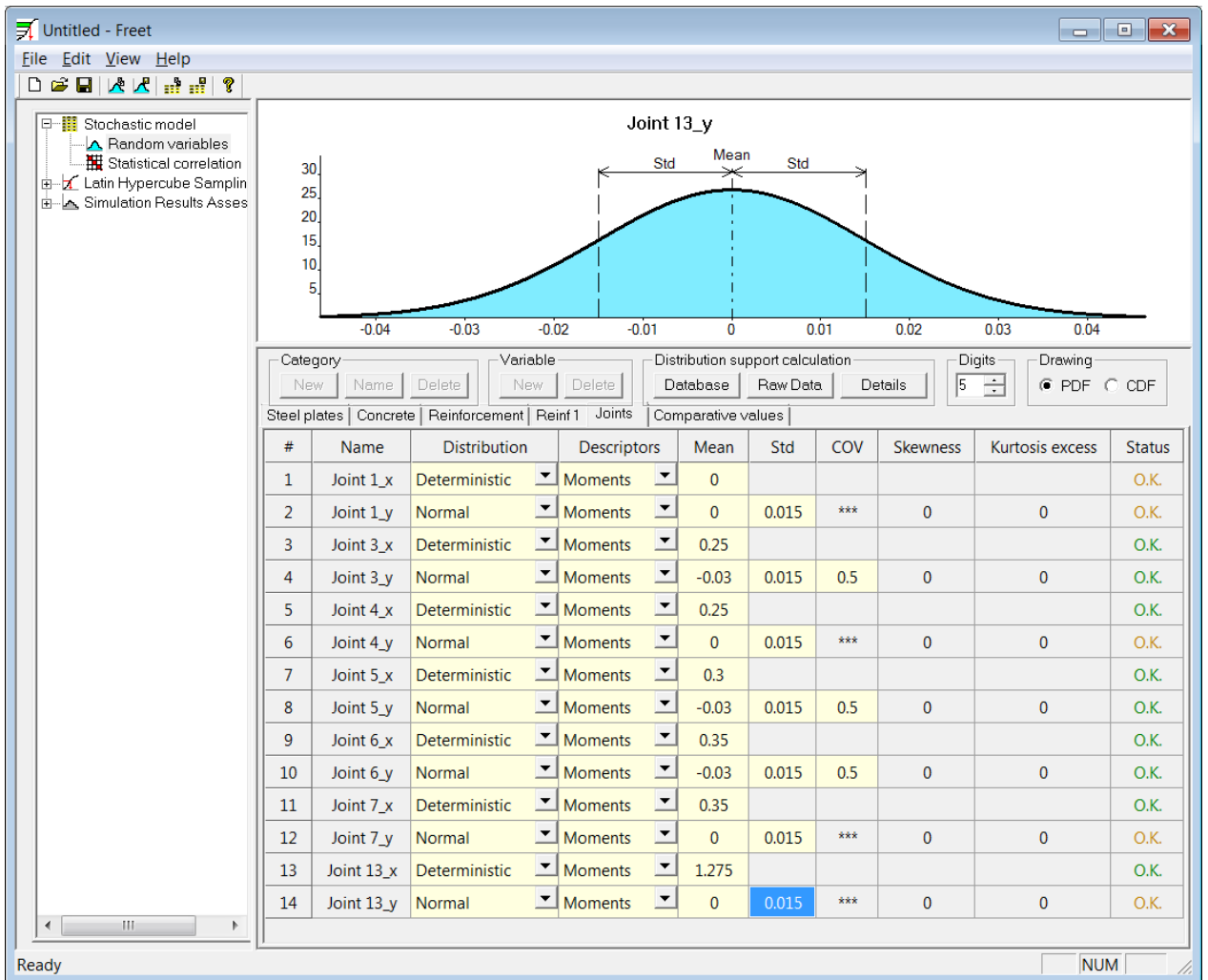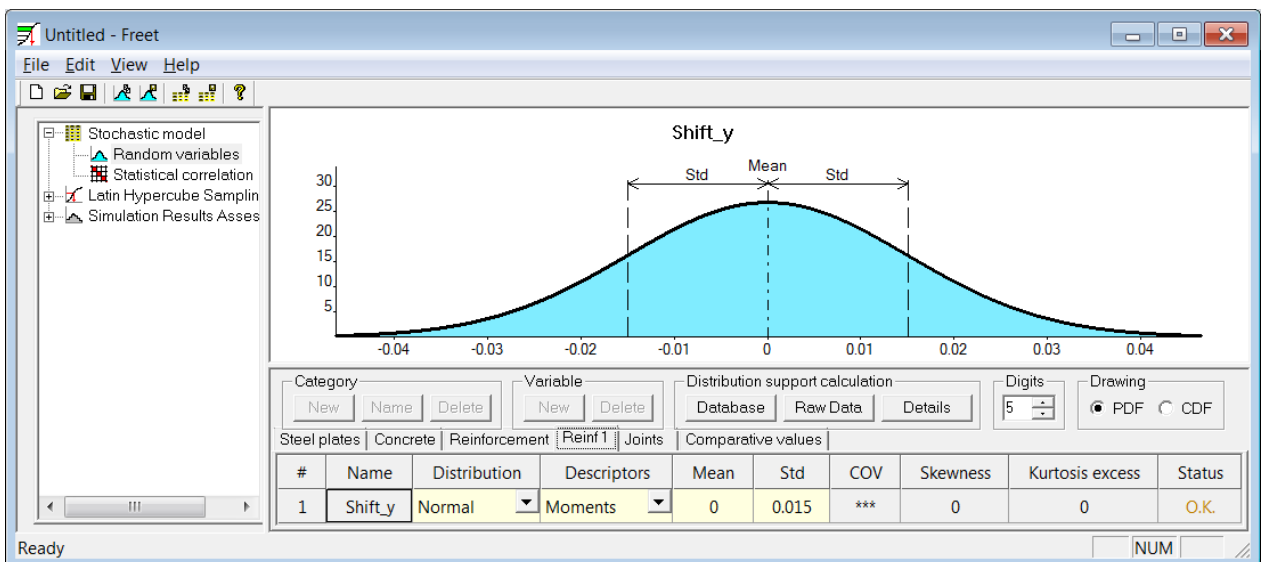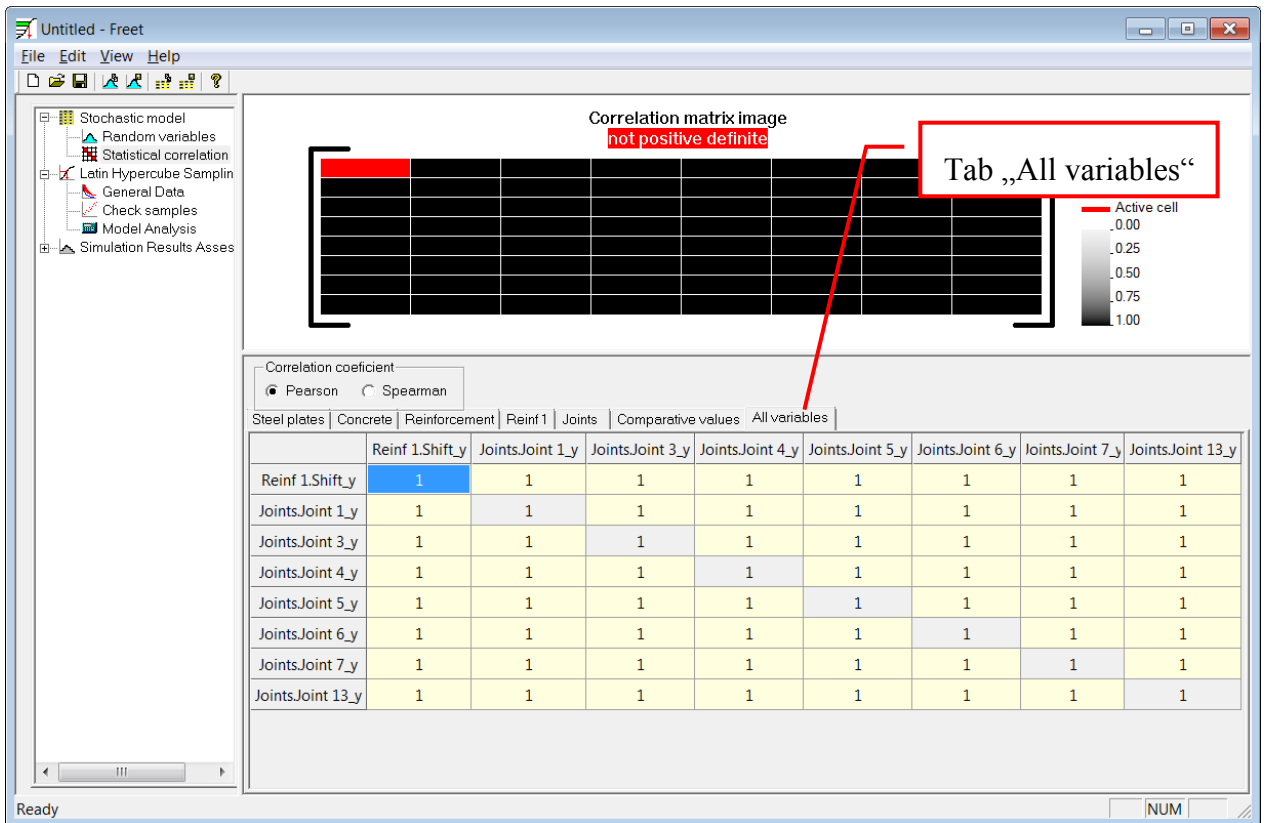**Figure 58 FReET – Correlation matrix of all variables**

Another example when correlation among all variables might be useful is when two types of concrete (different characteristic strength) are used in the same structure and when one knows, that the parameters of such concretes are not truly independent.

3.11 Random fields

Random fields can be efficiently used in problems, where the effect of changing material properties in the specimen is to be captured. Such fields will produce inhomogeneous material, which properties change with geometric coordinates.

Prior to running **RLACS Studio**, the deterministic basic task must be modified. Materials and material parameters which support random fields (in this version of **RLACS Studio**) are summarized in tables at the end of this section.

In the basic task, it is first necessary to create a new **Material With Random Fields** (Figure 59). In its definition, select the **Base material**, whose properties are to be changed with coordinates (Figure 60). Finally, in appropriate macroelements change the original material (**Cementitious**) to the newly generated one (**random fields**).
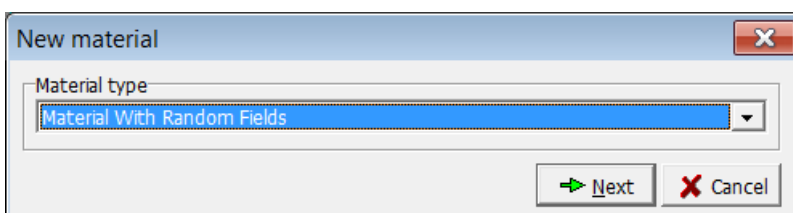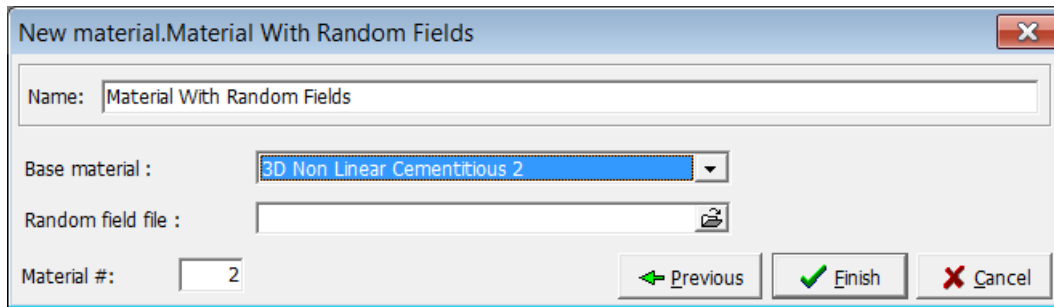

Figure 59 ATENA – new Material With Random Fields

Figure 60 ATENA – definition of Material With Random Fields

Afterwards, start **RLACS Studio**, create a new project and import basic task. In the tree menu in **Selector** expand **Materials** and check which parameters should be randomized. Notice, that there is the original name of the material (**Cementitious**) and not the material with random fields. Then click in the column labeled **Randomization type** to change the type **Variable** to **Field** (see Figure 61). If the randomization type is kept default (**Variable**), then the random fields are not used for that variable. Confirm your selection and then click **Randomize materials** or **Randomize inputs** to start **FReET**.
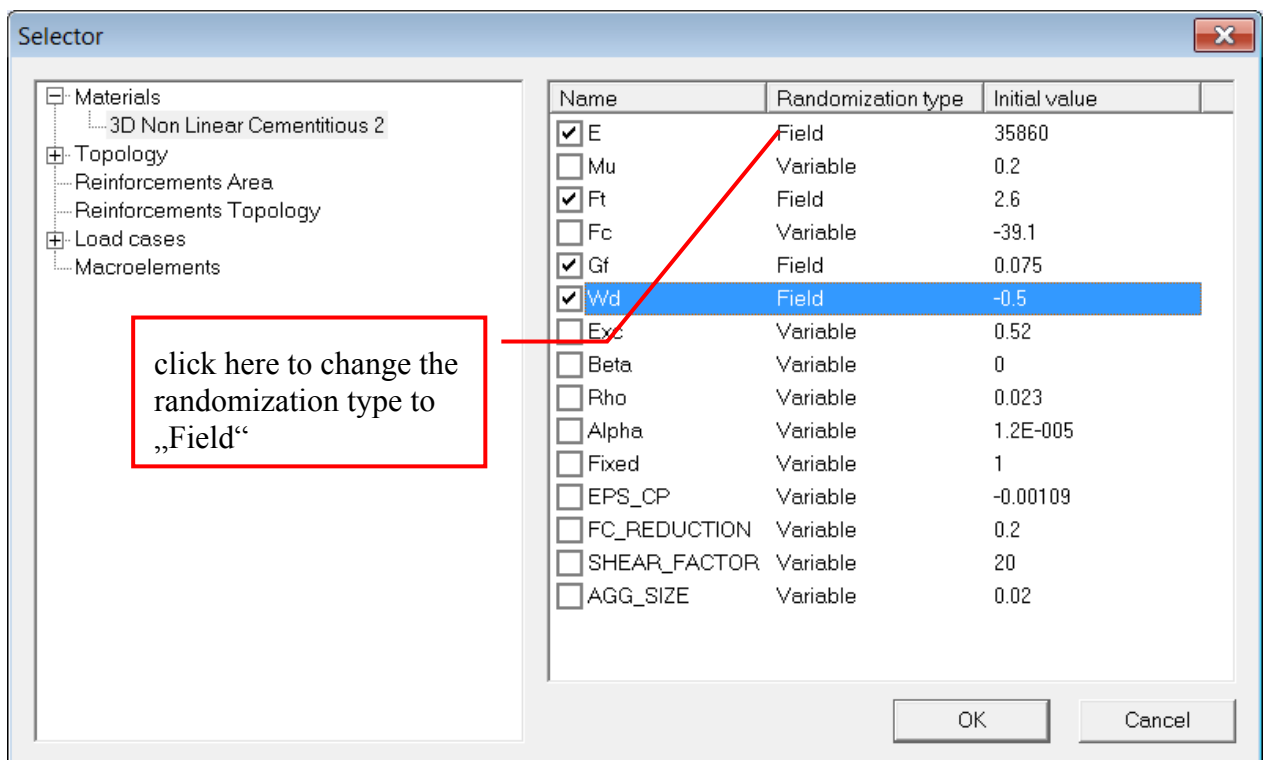
Figure 61 Selector – selection of random variables

In **FReET**, expand **Stochastic model** and then enter **Random Fields** (Figure 63). Proceed with definition of standard parameters (mean value and the standard deviation (or the coefficient of variation)) and then for every parameter choose value of *dx* and *dy,* which are the correlation lengths in the horizontal and vertical direction respectively.

Figure 62 shows distribution of tensile strength in the FE model if different correlation lengths are used. In this specific case correlation length 0.5 m was used in horizontal

and 0,1 m in vertical direction. This means that material properties differ 5x faster in vertical than in horizontal direction.

Step 1, Reinforced beam
Scalars:iso-areas, Basic material, in nodes, Random Field Values, Ft, <1.946E+00;2.701E+00>[None]
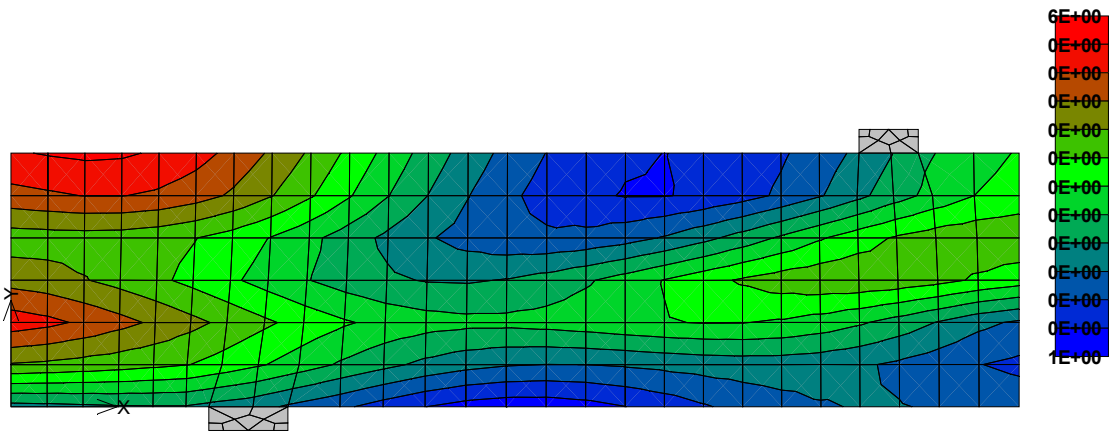
Figure 62 ATENA postprocessor – demonstration of different correlation lengths in horizontal and vertical direction

It is not possible to define correlation among the field variables. Continue to **Latin Hypercube Sampling | General Data** to generate desired number of simulations. To check generated fields, click **Latin Hypercube Sampling | Check Fields** (Figure 64). In the end, start the model analysis.

A different way to check the fields is using ATENA postprocessor. Open particular computed task in **ATENA**, go to postprocessor and select **Scalars | contour areas**. From the pull-down menu below choose **Random Field Values** and then select a material parameter from the list below. Figure 65 shows distribution of the tensile strength in the specimen. Note that at least one solution step must be computed in order to check random fields in postprocessor.
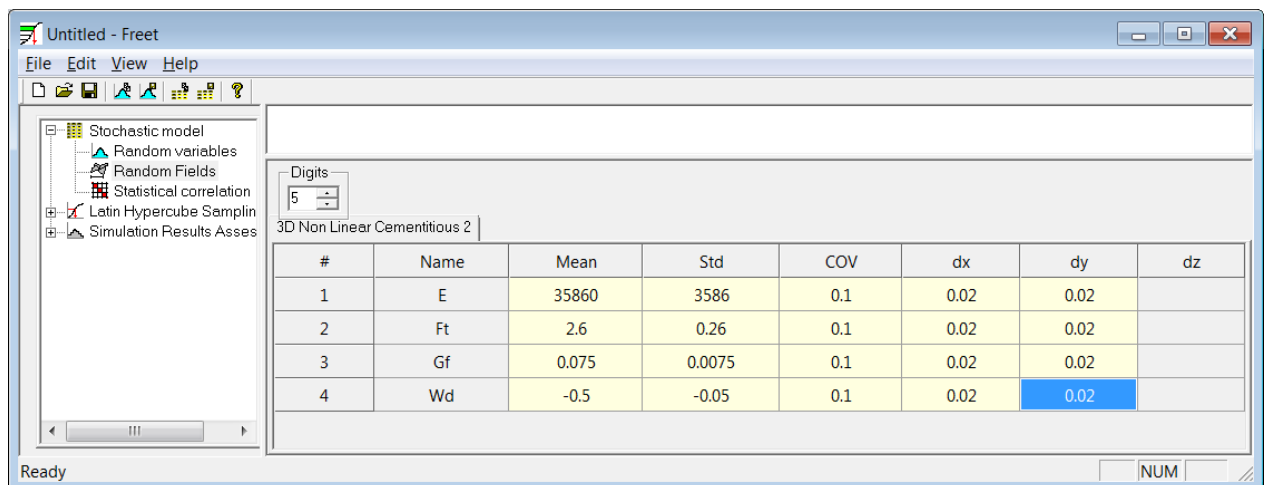
Figure 63 FReET – definition of random fields

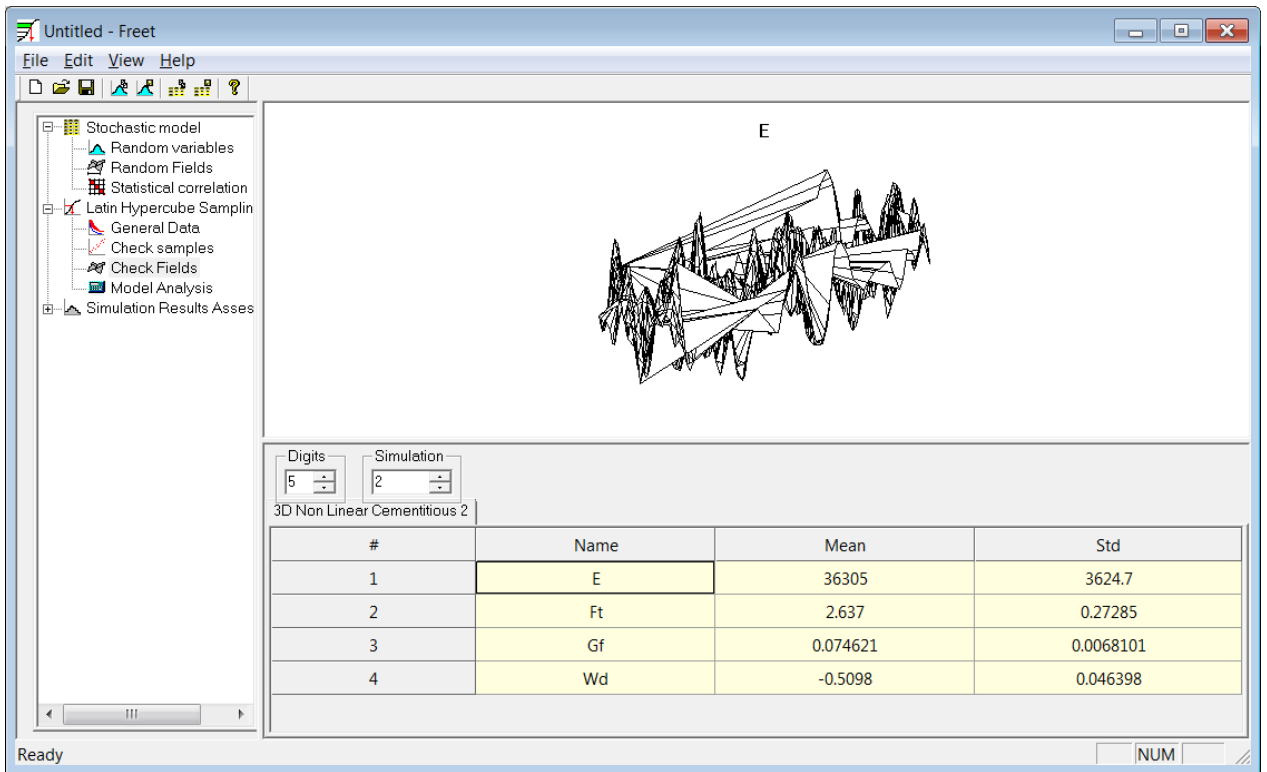**Figure 64 FReET – random field of the Young modulus in the second simulation**

Step 28, Zkoušky trámu
Scalars:iso-areas, Basic material, in nodes, Random Field Values, Ft, <2.009E+00;3.396E+00>[None]
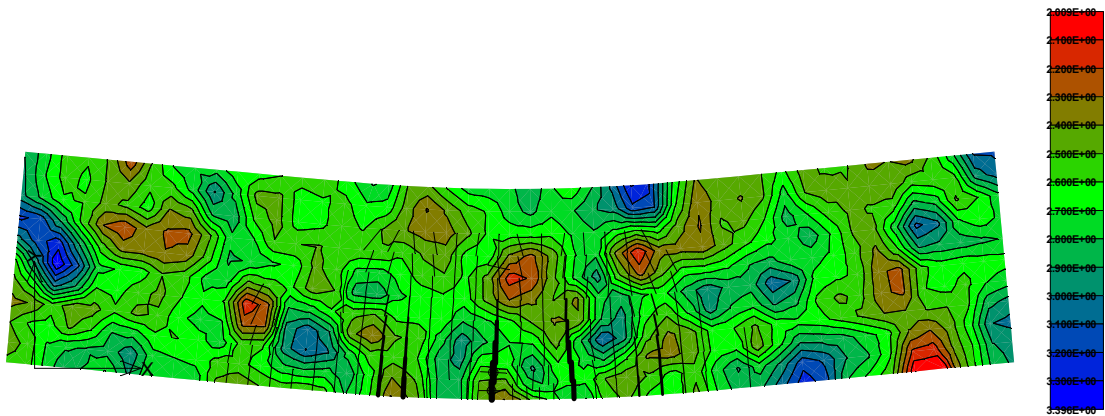Cracks: in elements, openning: <0.000E+00;7.620E-05>[m], Sigma_n: <-8.948E-02;3.176E+00>[MPa], Sigma_T: <4.344E-15;1.609E·

Figure 65 ATENA - postprocessor

```
FILETYPE ATENA_COORDINATES
FILESUBTYPE INTERNAL_POINTS_WITH_RANDOM_VALUES
VERSION 0

Group  Elem.   IP    Coord.(1)   Coord.(2)                               1.E                                    1.Ft
Units                        m            m  "3D Non Linear Cementitious 2".E  "3D Non Linear Cementitious 2".Ft
    1     1     1    0.0119000   0.1377000                   34176.207409117538                         2.859126025134
    1     1     2    0.0032000   0.1377000                   34695.016842606165                         2.989955633386
    1     1     3    0.0120000   0.1467000                   32700.885074793568                         2.818266009463
    1     1     4    0.0032000   0.1467000                   34746.974261909076                         2.891037082707
    1     2     1    0.0119000   0.1311000                   36747.978806387997                         2.764911550223
    1     2     2    0.0119000   0.1222000                   38471.743264335069                         2.724292024809
    1     2     3    0.0032000   0.1311000                   36616.366054045589                         2.945943845089
    1     2     4    0.0032000   0.1222000                   38386.057630285257                         2.886436181338
    1     3     1    0.0183000   0.1311000                   37301.634249082468                         2.552887093931
    1     3     2    0.0271000   0.1311000                   37424.542805452089                         2.454903148688
    1     3     3    0.0183000   0.1222000                   39199.791742805231                         2.582383508645
    1     3     4    0.0270000   0.1221000                   39147.386963396442                         2.506492919283
    1                                                        34390.960613324612                         2.657323733177
    1                                                        32255.985069990882                         2.672456850790
    1                                                        35294.927965717150                         2.530024020397
    1                                                        33757.393078674053                         2.523708013592
    1                                                        35400.490011248578                         2.626597335205
    1                                                        36049.008893545950                         2.596635197831
    1     5     3    0.0574000   0.1377000                                                              2.518051949164
    1     5     4    0.0575000   0.1467000                                                              2.584468710782
    1     6     1    0.0422000   0.1377000                                                              2.663654811341
    1     6     2    0.0335000   0.1377000                                                              2.590481784109
    1     6     3    0.0423000   0.1467000                                                              2.561324648178
    1     6     4    0.0335000   0.1467000                                                              2.512949524412
    1     7     1    0.0422000   0.1311000                                                              2.691895887908
    1     7     2    0.0422000   0.1221000                   35753.284026472640                         2.646297358071
    1     7     3    0.0335000   0.1311000                   35985.111763554603                         2.564414112935
```

Values of the Young modulus in IPs

Values of the tensile strength in IPs

Figure 66 ccp file for variable-field problems

Remark: *.ccp and *.ccm files

If the external fields are used, unique *.ccp and *.ccm files are generated for every simulation. Just one set of *.ccp and *.ccm files is generated if the ordinary randomization (randomization type = variable) is performed.

*.ccp file contains not only coordinates of integration points (column **IP**) of all elements, but also values of all field-dependent parameters in those points. See Figure 66.

*.ccm file contains description of used materials plus path of the corresponding *.ccp file, which contains the field description.

Remark: changing the field of random variables

The random field can be easily edited or changed.

To modify the random field open particular ***.ccp** file in an arbitrary text editor, perform changes and afterwards save that file.

In order to load a random field of a different task or to load a modified field, open the particular task in **ATENA**, then in the tree menu on the left side select **Materials**. Afterwards, double-click (or click Edit) on the appropriate material with random fields (Figure 67). Now, in the **Random field file** box the desired ***.ccp** file can be chosen.
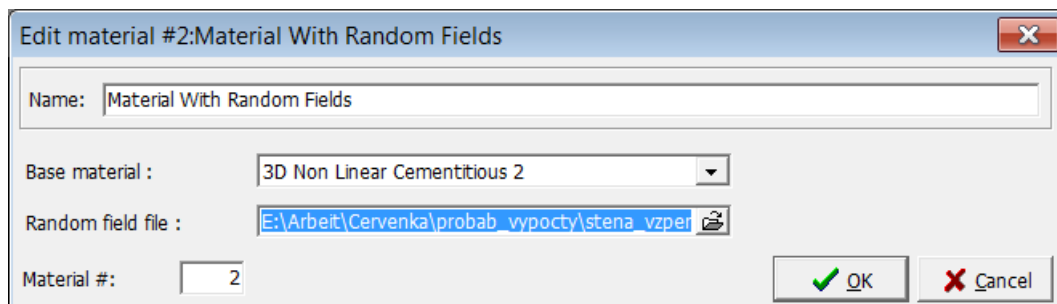
Figure 67 importing *.ccp file into ATENA

<u>**Remark: supported materials**</u>

Elastic materials

Material	E	μ	α_T	ρ
Plane Stress Elastic Isotropic	✓	✗	✓	✗
Plane Strain Elastic Isotropic	✓	✗	✓	✗
Axi Sym Elastic Isotropic	✓	✗	✓	✗

Plasticity-based materials

Material	E	μ	α_T	ρ	α_{DP}	K	w_d	β	Yield strength	Hardening modulus
Drucker-Prager	✓	✗	✓	✗	✓	✓	✓	✓		
Von Mises	✓	✓	✓	✗					✓	✓

Material models suitable for concrete

Material	SBeta	Non-linear Cementitious 2	Non-linear Cementitious 2 User
E	~	✓	✓
μ	✘	✓	✓
α_T	✘	✓	~
ρ	✘	✘	✘
fc	✓	~	~
ft	✘	✓	✓
Gf	✘	✓	
ε_{CP} or ε_C	✘	✓	
w_d	✘	~	
red. of compr. strength due to cracks	✘	~	
s_F		✓	
aggr. size		✓	
Exc		✓	~
β		✘	✘
fixed crack		✓	✘
Tension characteristic size			✓
Tension strain at localization			✓
Compression characteristic size			~
Compr. strain at localization			~
Shear strain at localization			~

3.12 Handy tool – calculator in FREET

Program **FREET** cannot be started directly, only from **RLACS Studio**. Expand **Stochastic model | Random variables**, choose any material parameter and click on button **Details** afterwards. A dialog, such as in Figure 69 should appear.
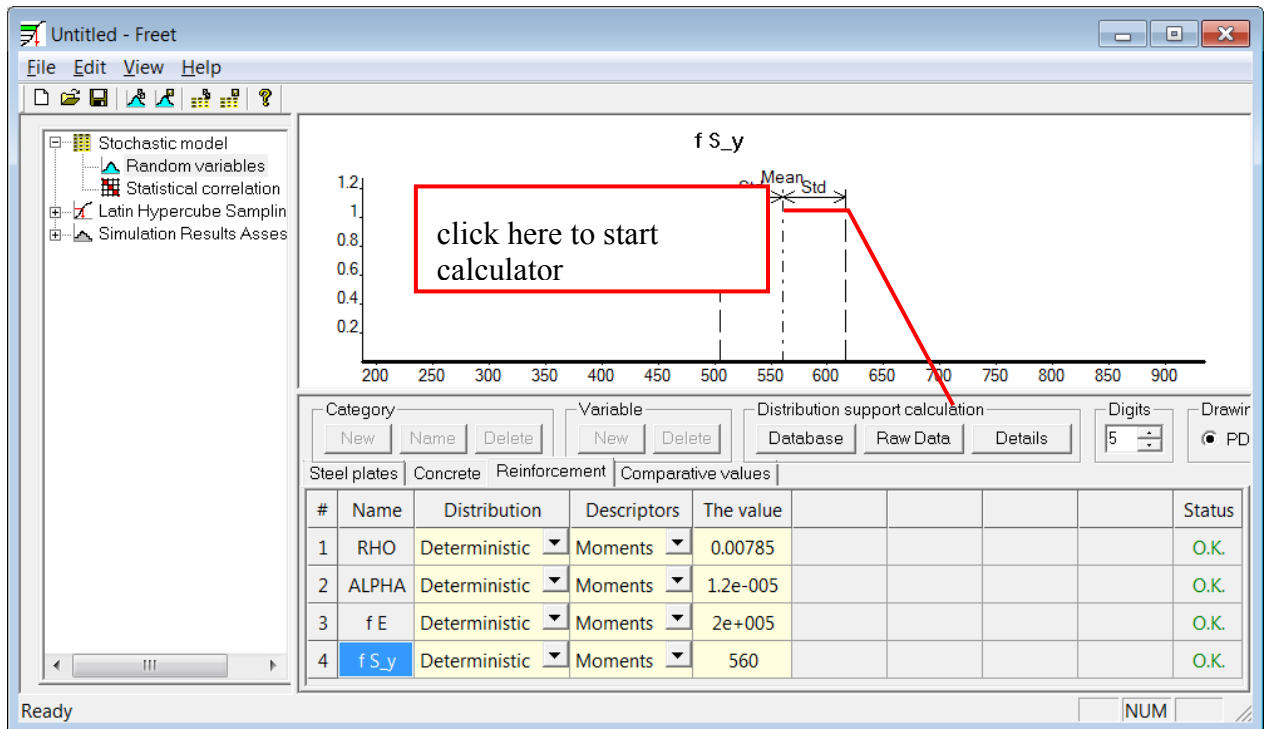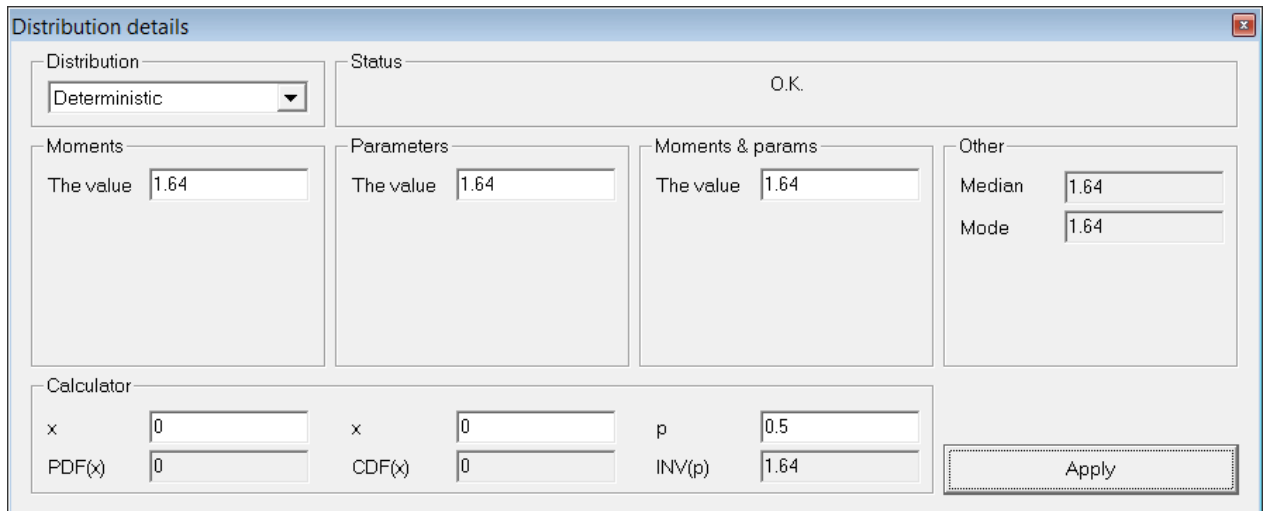

Figure 68 FREET – starting calculator

Figure 69 FREET – distribution details

3.12.1 Example #1 – definition of material properties from characteristic strength

Let's say that for reinforcing steel class B500 it is necessary for to define probabilistic distribution of its yield stress. One knows that its characteristic value of the yield stress, f_{yk}, is equal to 500 MPa, which corresponds to 5% percentile of its pdf.

For example if the coefficient of variation was 5% and the yield stress had normal distribution, then using a simple iterative procedure one finds that the mean value is 544.81MPa.
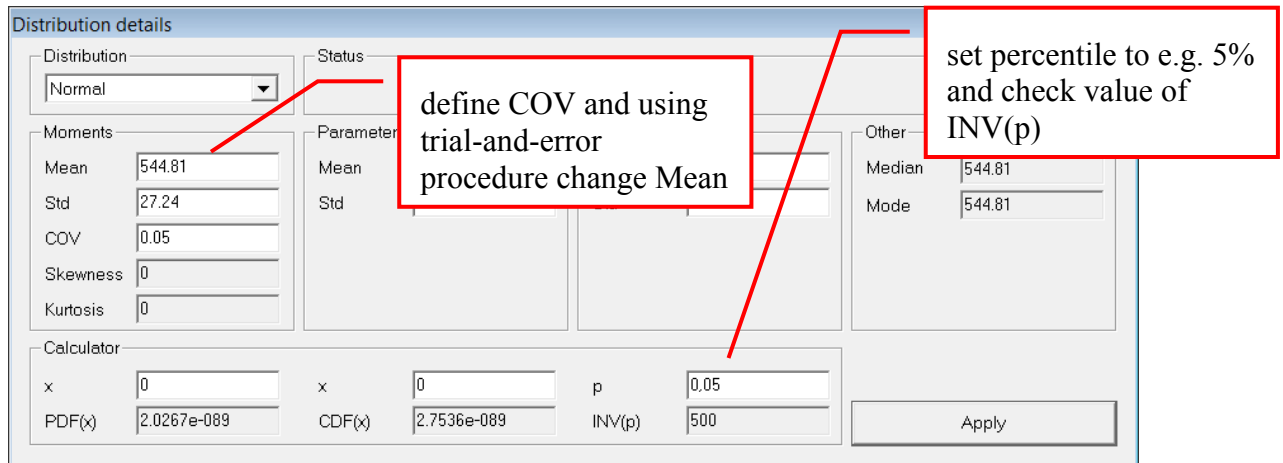
Figure 70 Finding mean value

Click on a button **Apply** to replace probabilistic distribution of a currently selected material parameter for those, which have been just generated.

3.12.2 Example #2 – application of the probability density function

Define type of distribution and its parameters. Let's that for steel from previous example it is necessary to find value of PDF(500 MPa) . Following two figures show the procedure and graphical explanation.
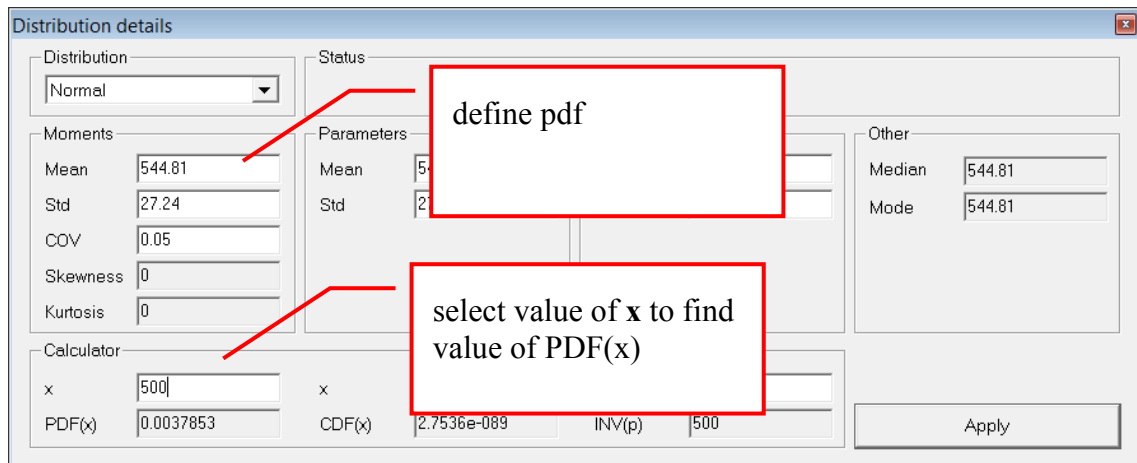

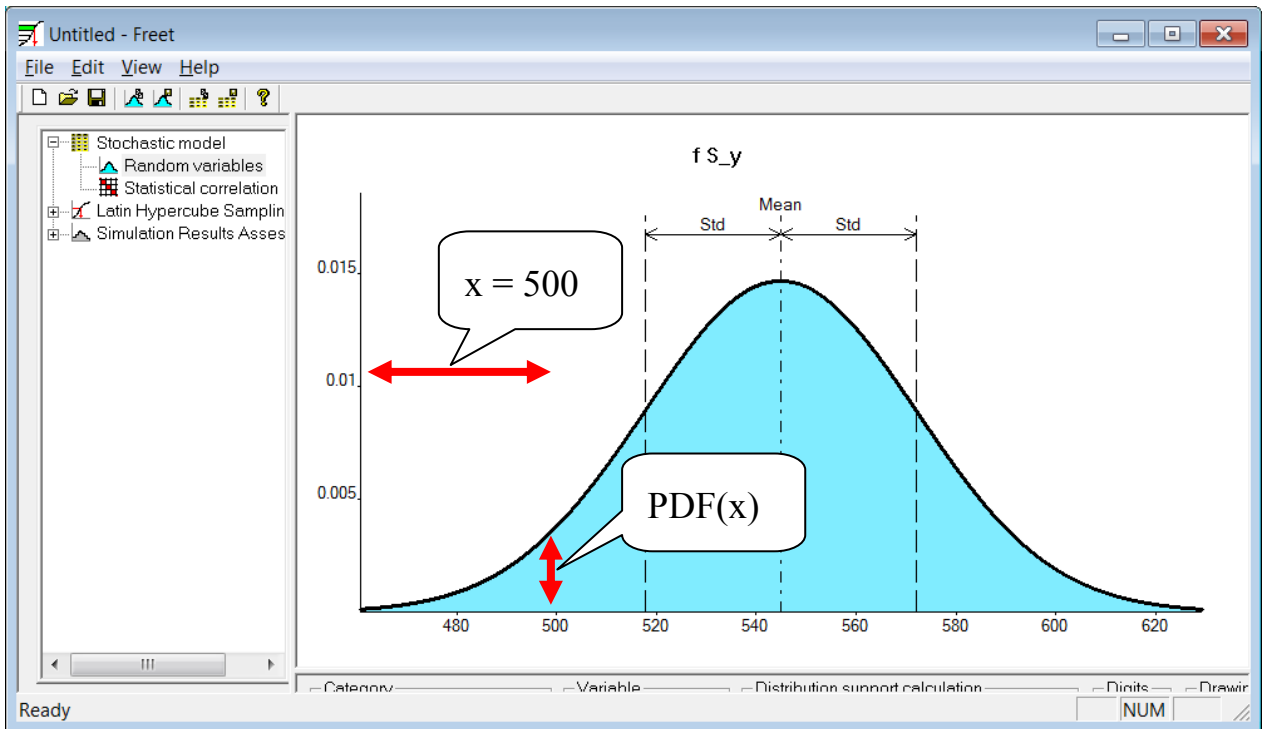**Figure 71 Finding particular value of a PDF**

Figure 72 Finding particular value of a PDF

3.12.3 Example #3 – application of the cumulative distribution function

This function enables to:

- for a specified value find the percentile (i.e. to evaluate the CDF); or
- for a given percentile to find the inverse value of a CDF

Following figure shows both for steel from Example #1 and p = 30% = 0.30.

These two functions can be extremely useful when one needs to find 5% percentile for material properties or 95% percentile for load effects.
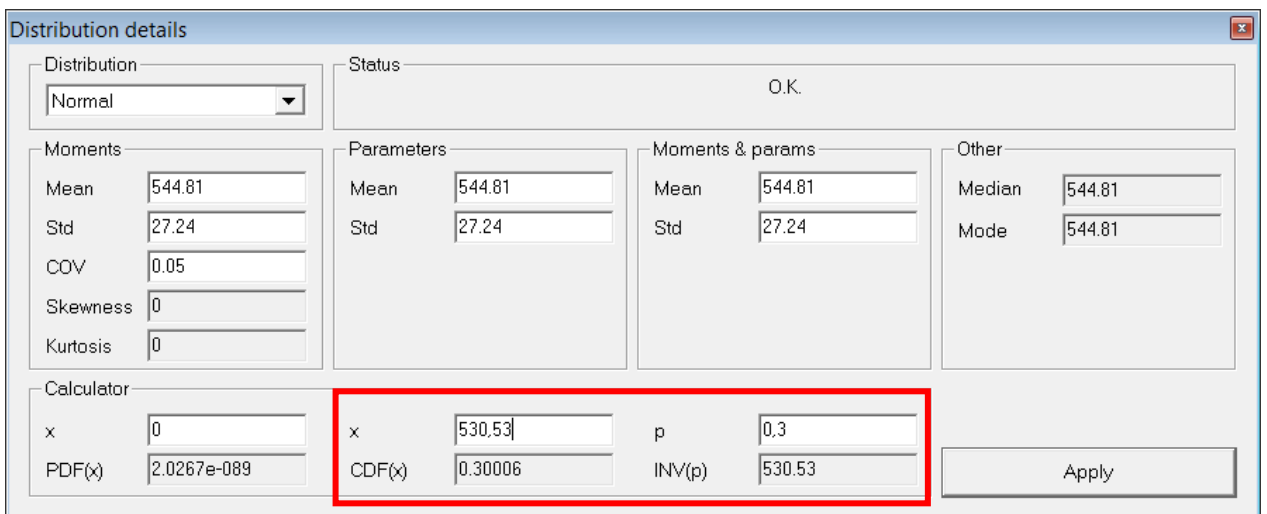
Figure 73 Evaluating CDF(x) and the inverse operation
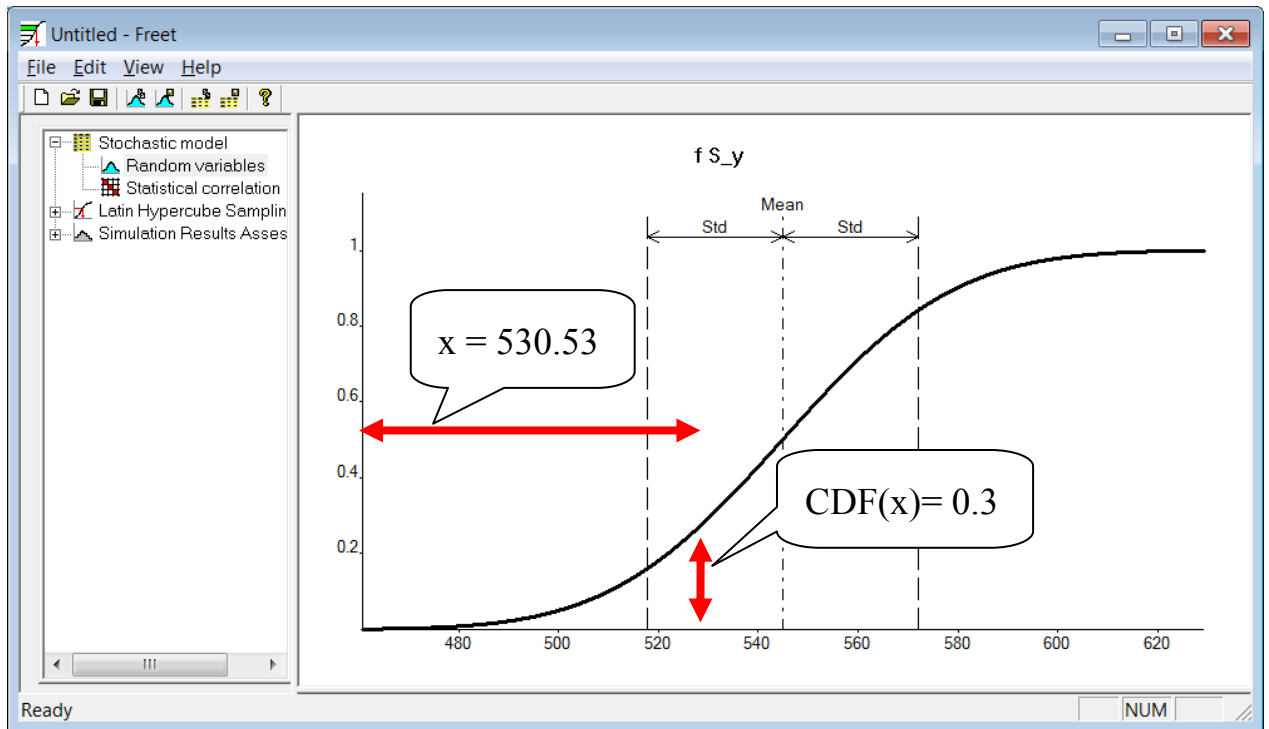
Figure 74 Evaluating CDF(x) and the inverse operation

3.12.4 Example #4 – finding the probabilistic distribution based on experimental data

Expand **Stochastic model | Random variables**, choose any material parameter and click on button **Raw data** afterwards. Fill the dialog with colon or space separated values. For more precise values use a decimal point (.), not a comma (,). It is also possible to load data from a text file.

When you are finished, click on button **Calculate parameters**. In an instant, most suitable type of distribution, mean value, standard deviation and other parameters appear. It is possible to change the type of distribution. Click **Apply** to use generated distribution for a selected parameter.
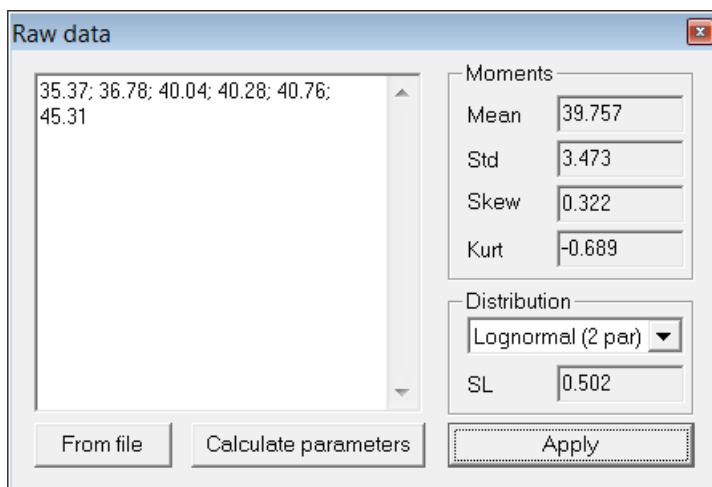

Figure 75 Determination of a type of distribution and its parameters from raw data

4 EXAMPLES

In all examples shown in this section use a simple cantilever loaded by its dead weight and force acting at its end. Following phenomena are modelled and presented here:

- influence of position of reinforcing bars (geometry of computational model is randomized)

- ultimate limit state is assessed

- serviceability limit state (only maximum deflection) is assessed

Consider a balcony cantilever with thickness of 160 mm, length 1600 mm and depth 1000 mm. Materials used in this simple structure are: concrete of the strength class C20/25 and steel B500 (rebars $\varnothing 8$ mm). Thickness of the covering layer is 30 mm. According to EuroCode 2 the necessary reinforcement area was computed as $5\varnothing 8$.

According to design standard Eurocode 1 this structure should withstand its dead weight and force F = 2 kN acting at the end. Considering all safety quotients the design moment M_{Ed} was calculated as 11,712 kNm, whereas the design resistance of the cross section M_{Rd} is 13.4 kNm.

A deterministic model was created in **ATENA 2D**. As shown in Figure 76, it consists of three macroelements; list of materials is shown below. On the leftmost edge of the first macroelement horizontal constraints are prescribed. The model is vertically supported only at the bottom-left corner.

Comments on the computational model

- The computational model consists of 8 nodes, 10 lines, 3 macroelements and 1 reinforcing bar. FE model contains 988 nodes and 796 finite elements. Totally 4 load cases are assumed (1 for supports, 2 LC prescribing forces and one prescribing deflection at the end of the cantilever). Two monitoring points are used: the first one measures deflection at the end of the cantilever and the second one measures vertical deflection at the bottom left corner.

- For the first macroelement (the leftmost) linearly elastic and isotropic material with stiffness of steel was chosen. It is used to guarantee that the monitoring point measuring vertical reaction will be linked to the same node even if the geometry is changed. The cantilever is vertically supported just at one point (in order to measure the whole vertical reaction) which leads to high stress concentration; usage of a different material model would have led to cracking or plastic deformations. This macroelement is very stiff; therefore the deflection of the end of this cantilever is not affected.

- The second (middle) macroelement describes the non-linear behaviour of concrete. To correctly capture all phenomena and cracking, very fine FE mesh (12 rows of quadrilaterals per height) is used.

- The rightmost (third) macroelement is made of linear elastic isotropic material with Young's modulus and Poisson's ratio of concrete. Quite coarse FE mesh (four rows of quadrilaterals per height) is used in order to speed-up computational performance. Thanks to linear elastic material it is not necessary to model the loading plate at the end of the cantilever.

- In order to capture real probabilistic behaviour of the structure, it is necessary to work with **mean** values of material parameters; concrete properties (for "mean values of C25/30") were generated in ATENA 3D material generator, and were manually copied to **ATENA 2D** afterwards. Mean value of reinforcing steel yield strength was estimated using FReET calculator (explained later) from covariance and characteristic yield strength $f_{yk} = 500$ MPa.
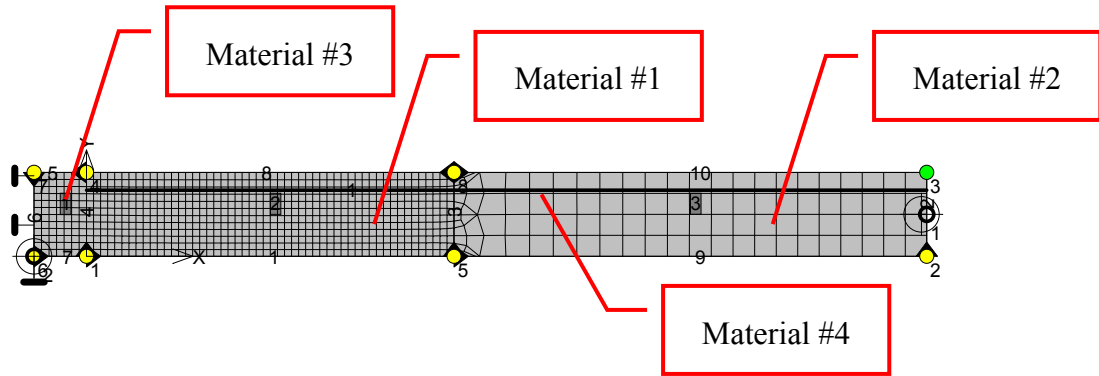
Figure 76 FE model of a balcony cantilever

```
Material n. 1
 Name : concrete C25/30 mean values - 3D Non Linear Cementitious 2
 Type: CC3DNonLinCementitious2
 Elastic modulus E = 3.100E+04 [MPa]
 Poisson''s ratio sm = 0.200 [-]
 Tensile strength F_t = 2.600E+00 [MPa]
 Compressive strength F_c = -3.300E+01 [MPa]
 Specific fracture energy G_f = 8.368E-05 [MN/m]
 Critical compressive displacement Wd = -1.0350E-03 [m]
 Eccentricity, defining the shape of the failure surface Exc = 0.520 [-]
 Multiplier for the direction of the plastic flow Beta = 0.000 [-]
 Specific material weight Rho = 2.300E-02 [MN/m3]
 Coefficient of thermal expansion Alpha = 1.200E-05 [1/K]
 Fixed smeared crack model will be used Fixed = 1.000 [-]
 Plastic strain at compressive strength EPS_CP = -5.000E-04 [-]
 Reduction of comp. strength due to cracks fc_LIM = 0.2 [-]
 Crack shear stiff. factor s_F = 20.0 [-]
 Aggregate size = 0.0200 [m]

Material n. 2
 Name : concrete elastic
 Type: CCPlaneStressElastIsotropic
 Elastic modulus E = 3.100E+04 [MPa]
 Poisson''s ratio sm = 0.200 [-]
 Specific material weight Rho = 2.300E-02 [MN/m3]
 Coefficient of thermal expansion Alpha = 1.200E-05 [1/K]

Material n. 3
 Name : steel elastic
 Type: CCPlaneStressElastIsotropic
 Elastic modulus E = 2.100E+05 [MPa]
 Poisson''s ratio sm = 0.300 [-]
 Specific material weight Rho = 7.800E-02 [MN/m3]
 Coefficient of thermal expansion Alpha = 1.200E-05 [1/K]

Material n. 4
 Name : rebars mean
 Type: CCReinforcement
 Typ: BiLinear
 Elastic modulus E = 2.000E+05 [MPa]
```

```
Sigma Y = 545.000 [MPa]
Specific material weight RHO = 7.850E-02 [MN/m3]
Coefficient of thermal expansion ALPHA = 1.200E-05 [1/K]

Active in compression
```

```
Sigma Y = 545.000 [MPa]
Specific material weight RHO = 7.850E-02 [MN/m3]
Coefficient of thermal expansion ALPHA = 1.200E-05 [1/K]
```

4.1 Influence of carrying capacity on position of rebar

4.1.1 Scope

Position of reinforcing bars is often subjected to statistical scatter; this example investigates influence of carrying capacity of a simple cantilever on vertical position of reinforcing bar.

The distance between the axes of rebars and the top surface of the cantilever is in this example assumed to change between 0 (see Figure 77) and 68 mm (double of the mean value; see Figure 78). In order to examine various combinations, rectangular distribution with these limits will be used.

Two problems are investigated. The second one differs only in area of reinforcing steel, which is in this case doubled.

There are two ways to model this problem. Either a reinforcing bar can be shifted or all joints of the cantilever. Presented simulation will show the latter possibility.
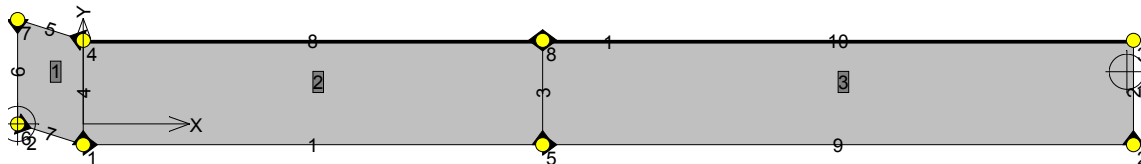
Figure 77 Lower bound of displacement – rebars almost coincide with the top cantilever surface
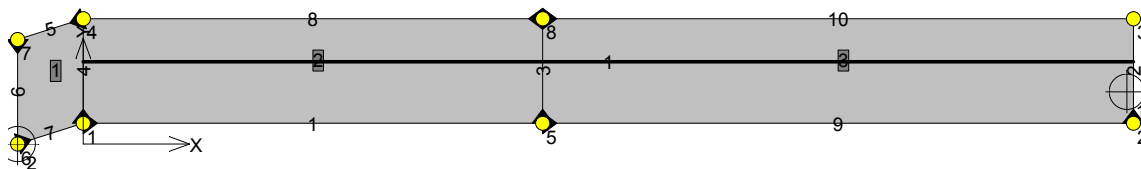
Figure 78 Upper bound of displacement – distance between the axis of rebar and the top surface of the cantilever doubles

4.1.2 RLACS project outline:

- Create a new project in RLACS studio, select balcony_rebar_position_1.cc2 as the basic task.

- In selector uncheck all materials, then in the tree menu expand **Topology** and then select all joints with non-negative x coordinate. Afterwards click **Randomize inputs ...**
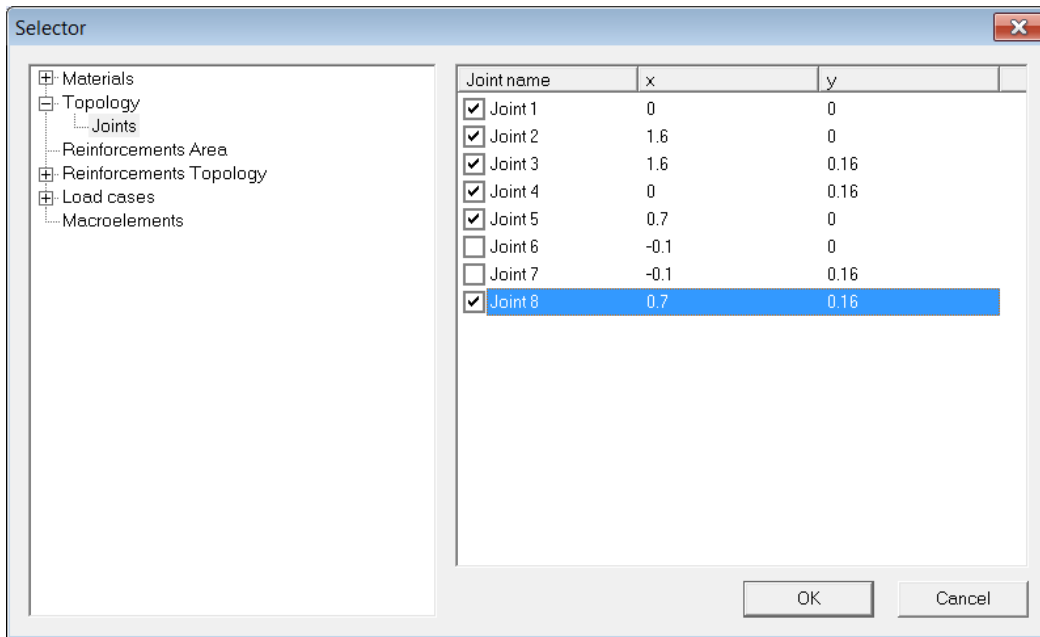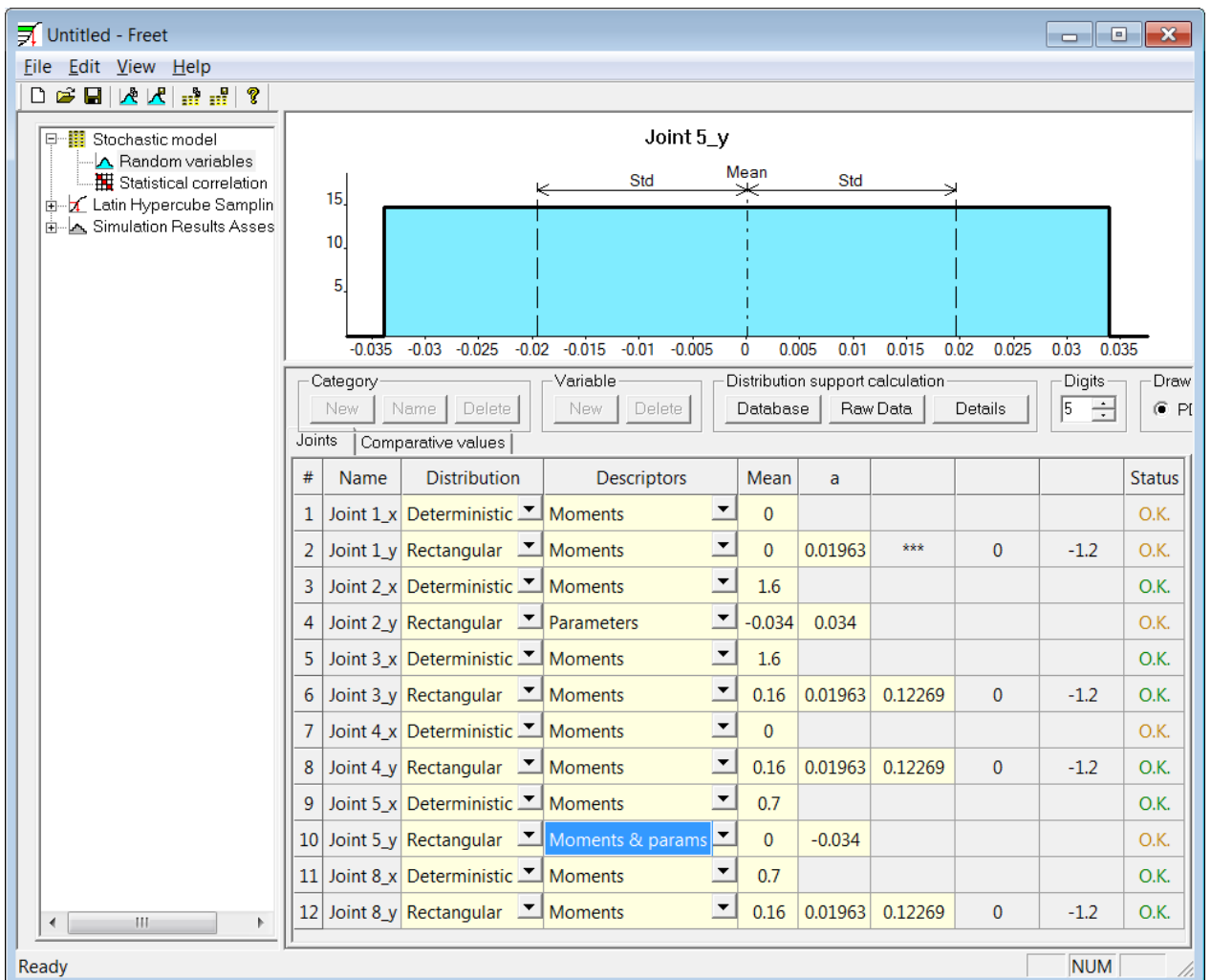
Figure 79 Joints selection

Figure 80 Probabilistic distribution of joints

- For every y coordinate specify its probabilistic distribution; the limits of movement should be same and in the range (-0.034, 0.034). The lower bound must not be exceeded to avoid that the rebar moves away from concrete (this would have led to singular stiffness matrix). Each type of probabilistic distribution can be described by 3 equivalent descriptors:
  - **moments** (Figure 80, Joint 1_y – mean and std must be specified);
  - **parameters** (Figure 80, Joint 2_y – lower and upper bound must be specified);
  - **moments & parameters** (Figure 80, Joint 5_y – mean and lower bound must be specified).
- Fill in the correlation matrix with ones. This means that movement of one joint forces other joints to move the same way. For reference see Figure 81.
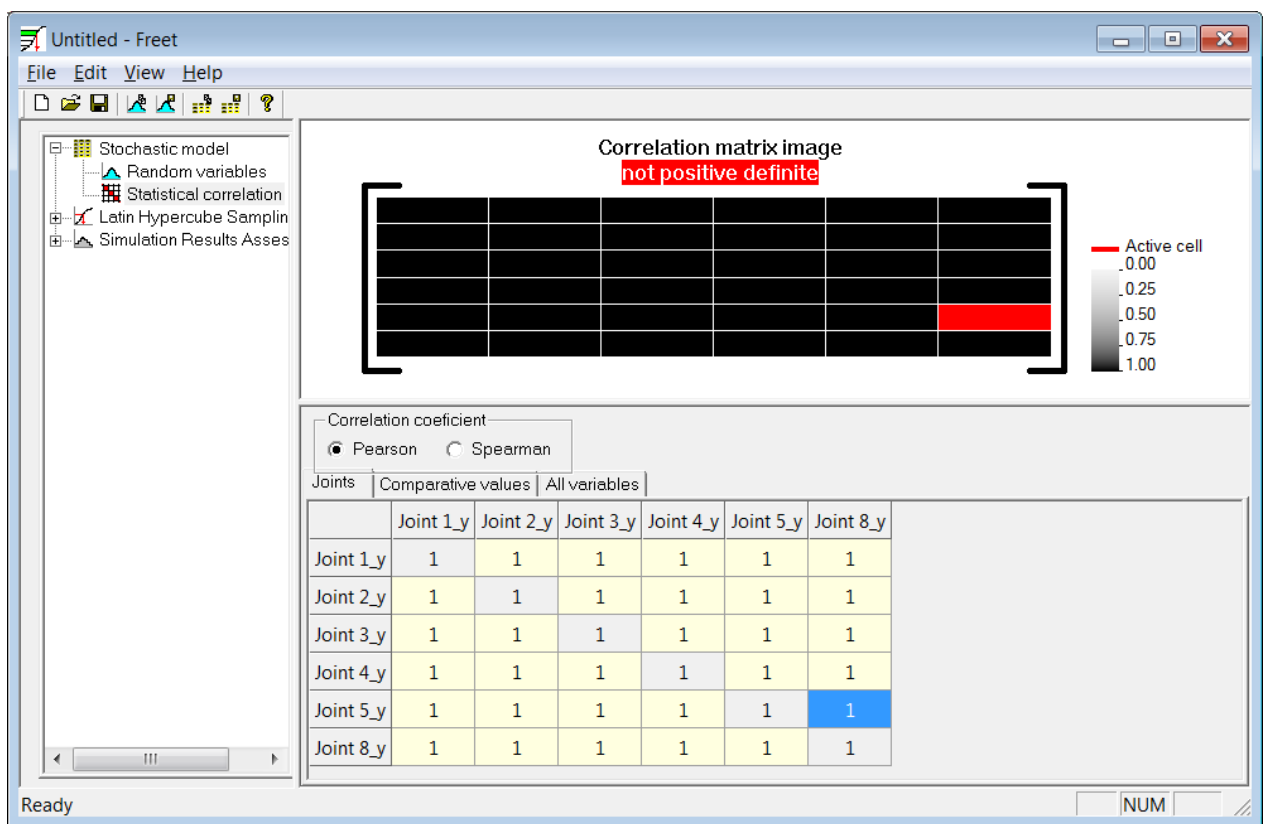
Figure 81 Definition of correlation matrix

- Generate random samples (e.g. 15).
- Check samples and then proceed to model analysis.
- Make sure that all tasks are finished. If not, go to specific job, refine analysis steps and import results to **RLACS** afterwards.
- Using **FReET** perform sensitivity analysis.

4.1.3 Remarks

Functionality of performed simulations can be easily verified:

- Lower number of simulation means lower value of displacement, which results in higher value of the lever arm of the internal forces (i.e. the distance between the resulting force of the compressed concrete and the axis of reinforcing bars). Hence, the lower the number of simulation the higher the resistance. See Figure 82)

- Higher content of reinforcement results in more straight shapes of LD diagrams. This means that the influence of the tensile behavior of concrete is reduced and the dominating factor becomes the reinforcement; compare Figure 82 and Figure 83).

- Sensitivity of the maximum carrying capacity to the displacement of reinforcing steel should be more pronounced in the second case (simulations with higher reinforcement content).
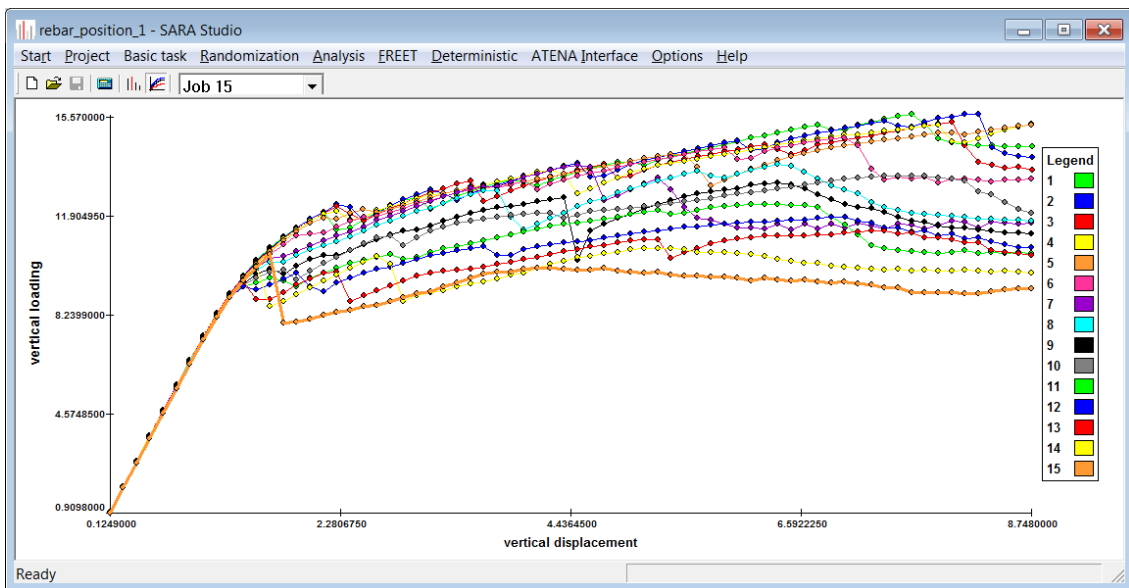
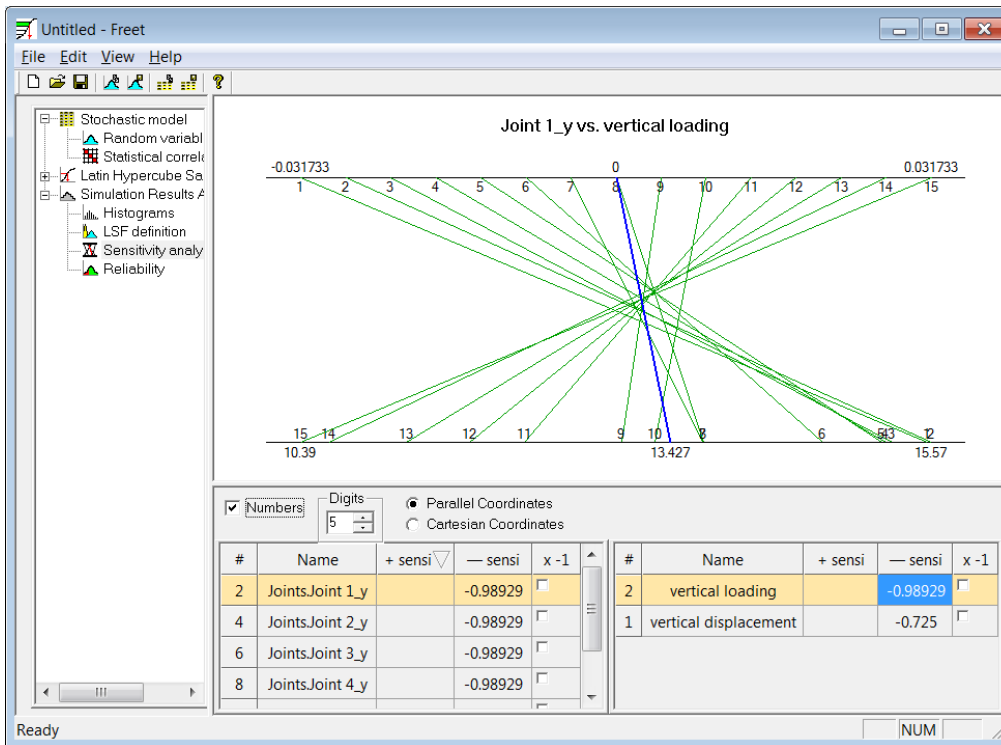**Figure 82 LD diagrams of 15 variants (5⌀8)**
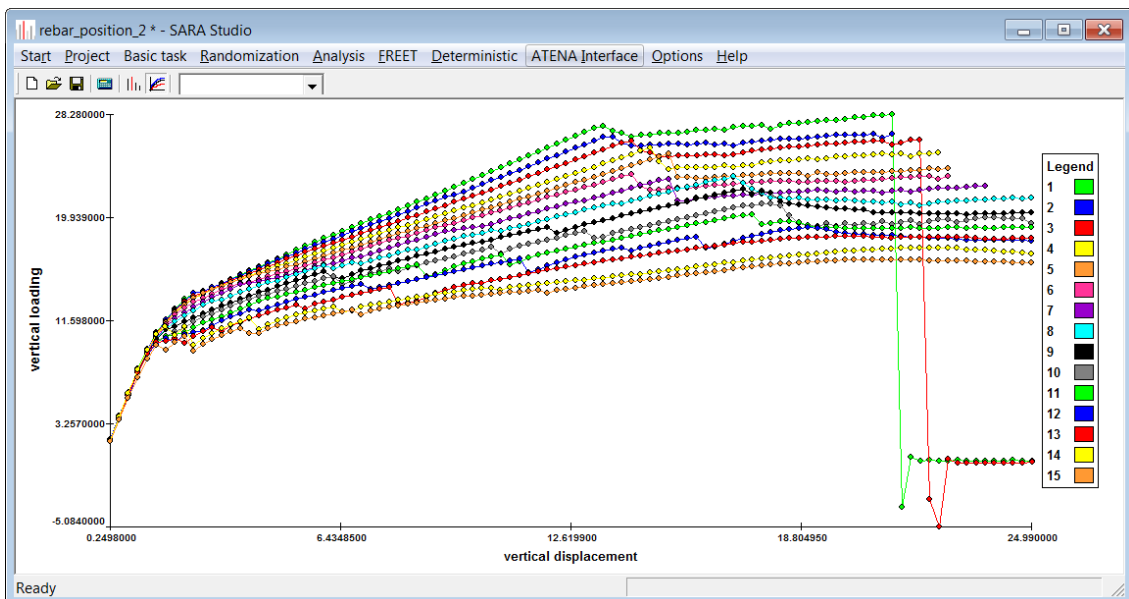
Figure 83 LD diagrams of 15 variants and additional rebars (10⌀8 in total)
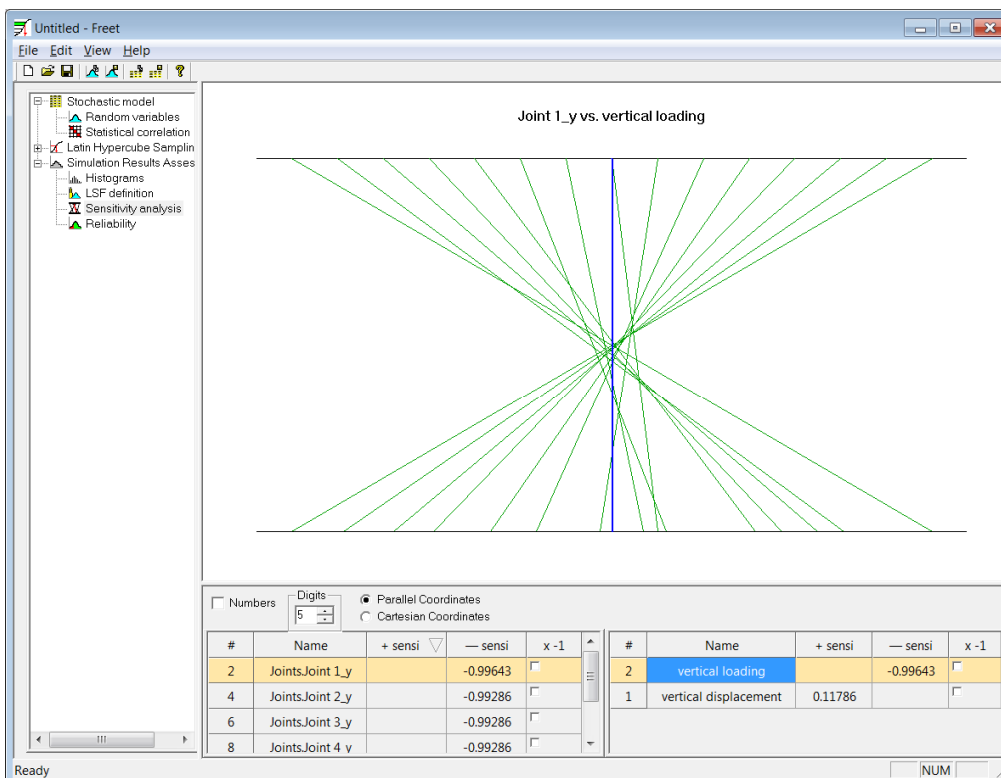
Figure 84 Negative sensitivity of maximum loading (carrying capacity of the cross section) and displacement of reinforcement. Above: simulations with 5∅8, sensitivity is -0.989; below: 10∅8, sensitivity is -0.996.

Presented example demonstrates correctness of assumptions used in currently valid design codes used for concrete structures. To investigate the actual probability of failure and the reliability index beta, more accurate (normal) distribution, which is closer to the reality, would have to be used.

4.2 Ultimate Carrying Capacity of a Cantilever

4.2.1   Scope

In this example, ultimate limit state is assessed. It is expected to obtain equal or lower reliability index beta than it is in the standard ČSN EN 1990. Only material properties of concrete and rebars are randomized in this case.

4.2.2   RLACS project outline:

- Create a new project in **RLACS studio**, select balcony_ULS_2.cc2 as the basic task.

- In **Selector** for following materials check:

  concrete C25/30 mean values: E, Ft, Fc, Gf and Wd

  concrete elastic: E (in randomized task it is supposed to be same as E in concrete C25/30)

  rebars mean: f S_y

  Other material parameters are assumed to be constant (deterministic).

- Click **Randomize materials …**

- According to following figure specify distribution properties for concrete C25/30
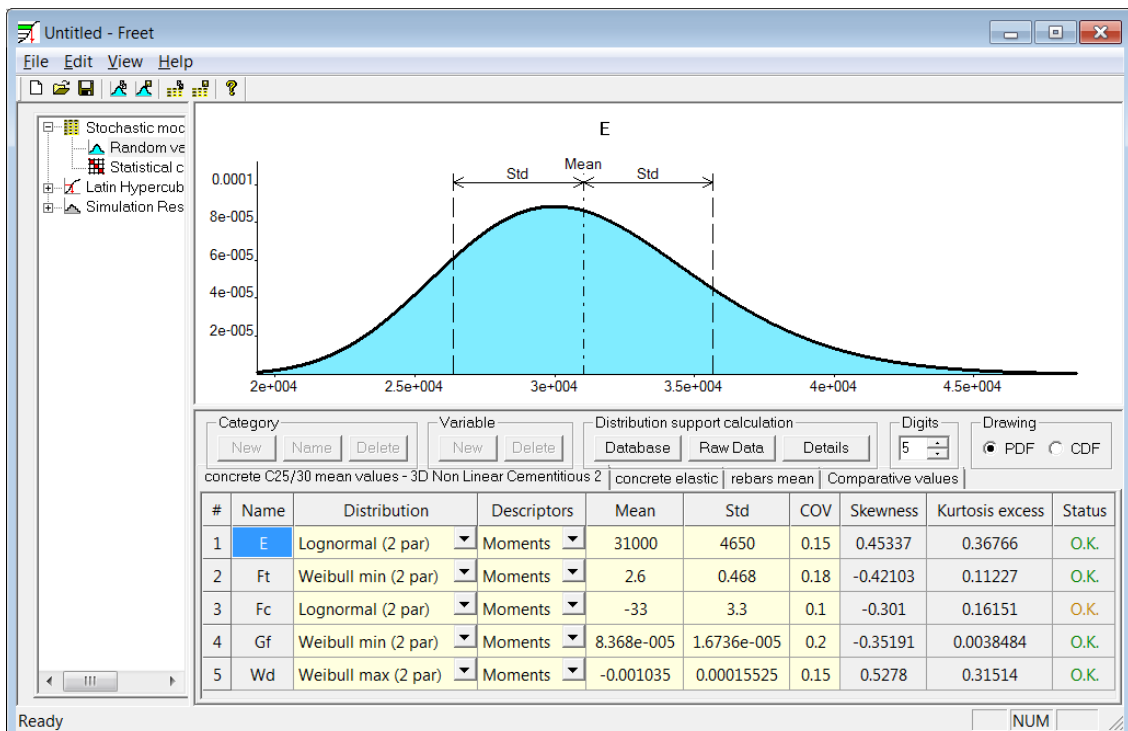
Figure 85 Probabilistic distribution for concrete C25/30 parameters

- For **concrete elastic** specify the same distribution as for E in concrete C25/30.

- For **rebars mean** specify lognormal distribution of pdf and COV set to 0.05. Click on button **Details** to check that the 5% quantile is equal approximately 500 MPa (see Figure 86). If one knows the characteristic value and the standard deviation or COV, the mean value can be easily computed this way. Button **Apply** serves for applying changes to selected material parameter.
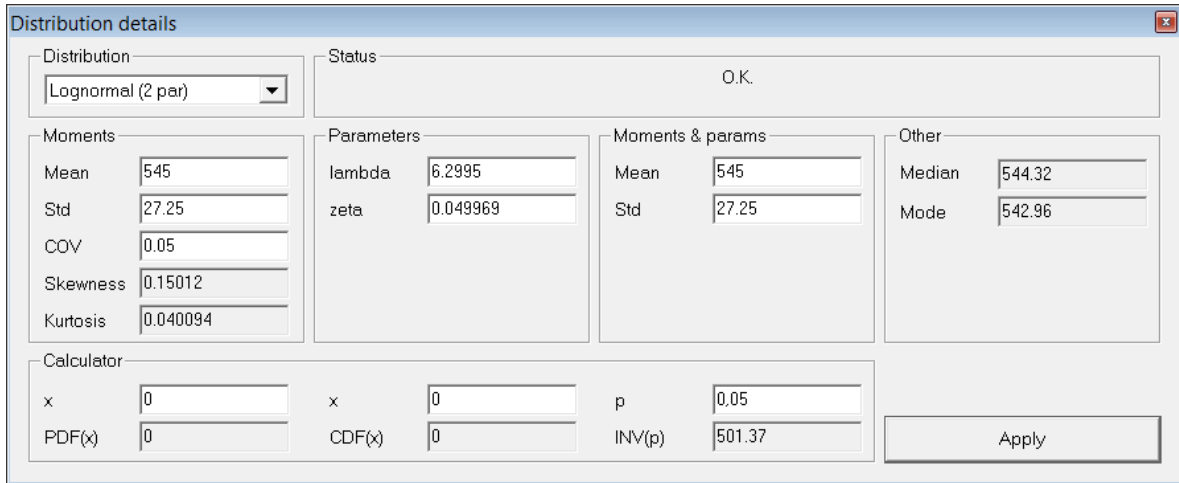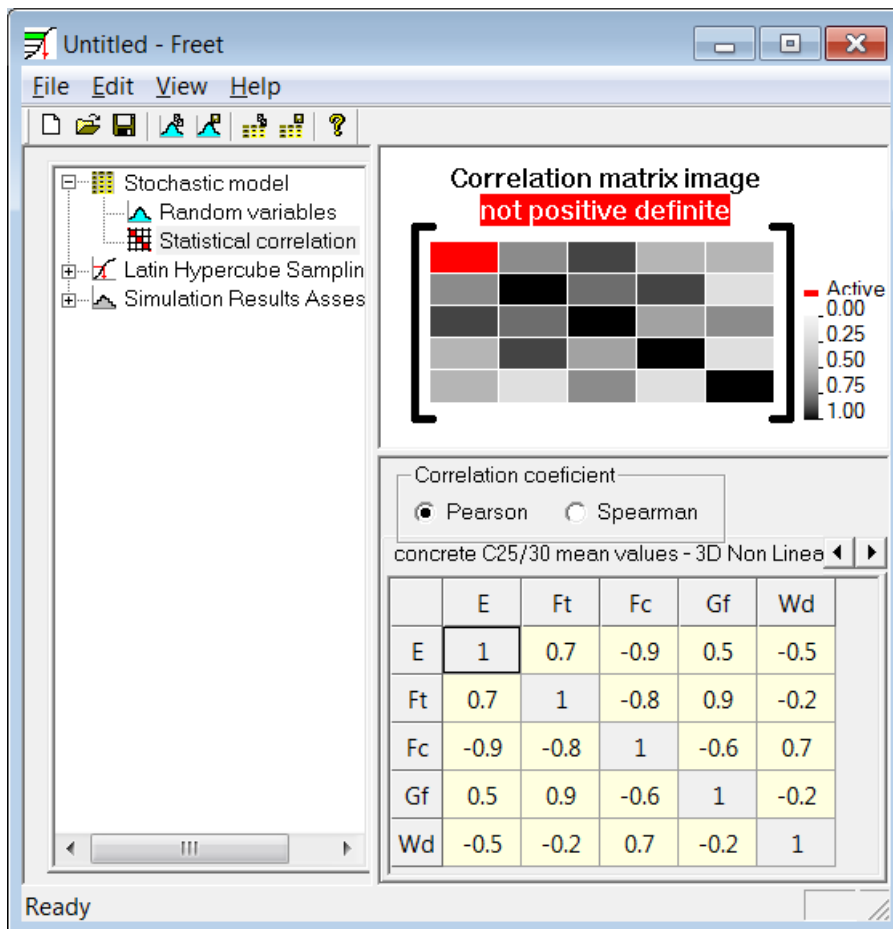

Figure 86 FReET – dialog for distribution details.

Figure 87 Definition of correlation matrix for material C25/30.

- Fill in the correlation matrix for material C25/30 (e.g. according to Figure 87). Note that negative quotients appear in cells, where the negative mean value of

one variable crosses positive mean value of a different variable. Still this expresses positive correlation.

- Yield strength of rebars is assumed to be independent of concrete properties, hence no correlation is defined.

- Elastic modulus of material **concrete elastic** must be in all randomized tasks fully linked to Elastic modulus in **concrete C25/30**. To do that, select tab **All variables** (still in **Statistical correlation**) and in cells where elastic moduli of these two material cross, put 1.0. Remaining cells on line **concrete elastic.E** by values from row **concrete C25/30 mean values**. (E.g. the value in cell **concrete elastic.E** x **concrete C25/30 mean values** is -0.9.)

- Create randomized samples (e.g. 15) and perform model analysis. One should obtain results comparable to Figure 88.
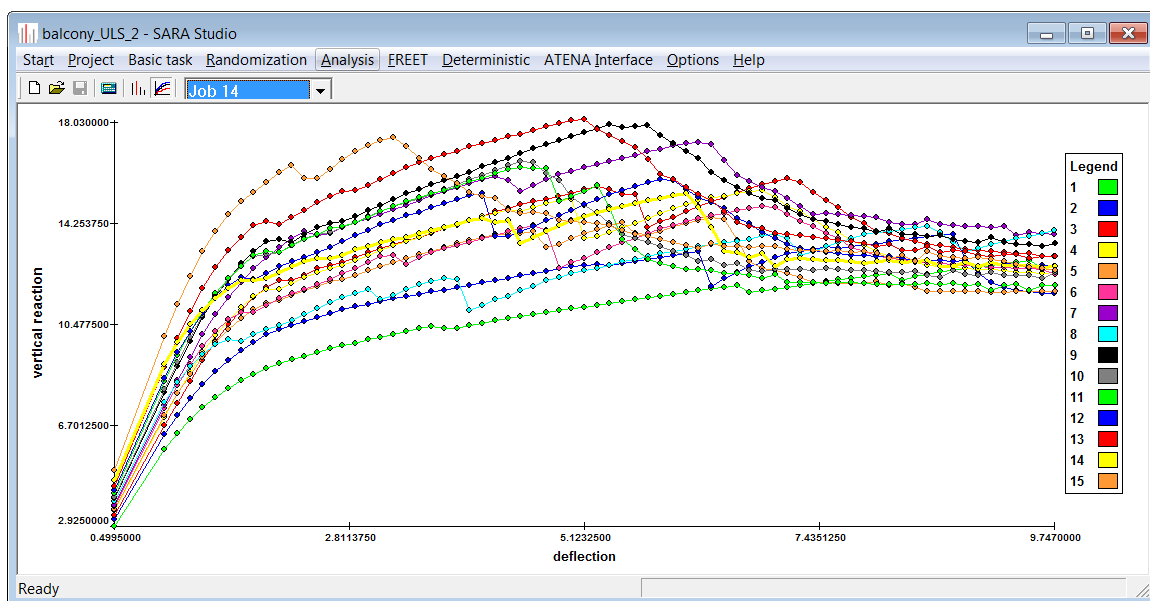

Figure 88 LD diagrams

- Click **FREET | Statistic evaluation**.

- In the tree menu on the left select **Stochastic model | Random variables** and then select tab **Comparative values**. Define a new comparative value named **Action** and using calculator (button details) define a normal distribution with 95% quantile equal to 8,4 kN. For reference see Figure 89.
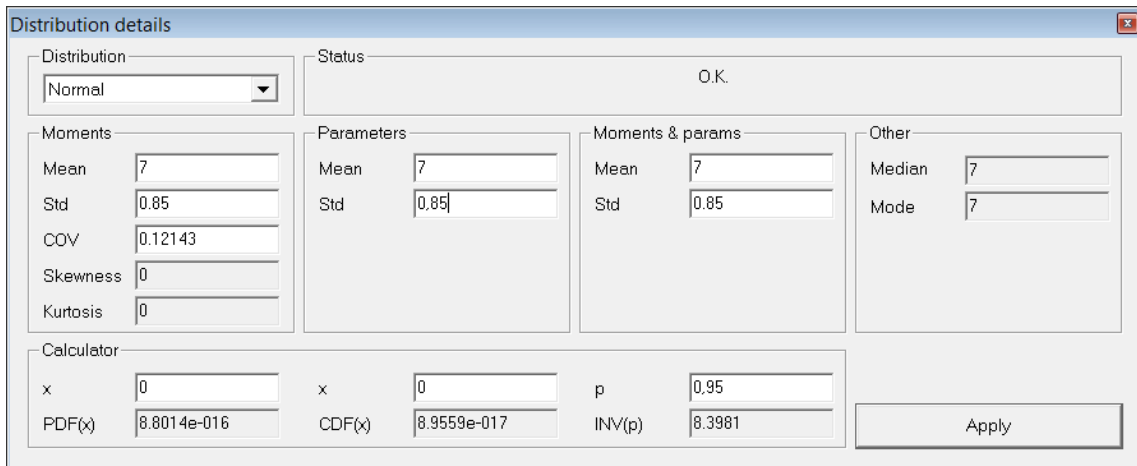
Figure 89 Distribution definition for comparative value named Action.

- Go to **Latin Hypercube Sampling | General Data** and click on button **Run**. Note that **Comparative values only** must be checked this time.

- Expand **Simulation Result Assessment | LSF definition** and click on button **New LSF**. Define a new LSF as a difference between the resistance of the structure (i.e. maxima of monitor "vertical reaction") and newly defined comparative value "Action". Result is shown in Figure 90.
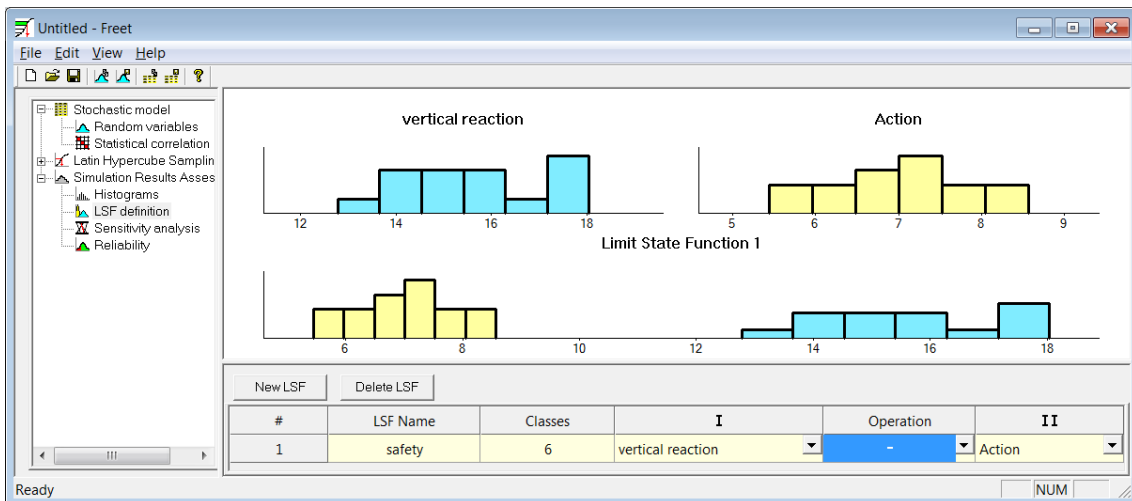

Figure 90 Definition of a new limit state function

- To assess the reliability, go to **Simulation Result Assessment | Reliability** and select 3rd row **safety**. The probability of failure is 2.54e-7 and the reliability index beta is 5.02, which is far more than the requirement in standards.
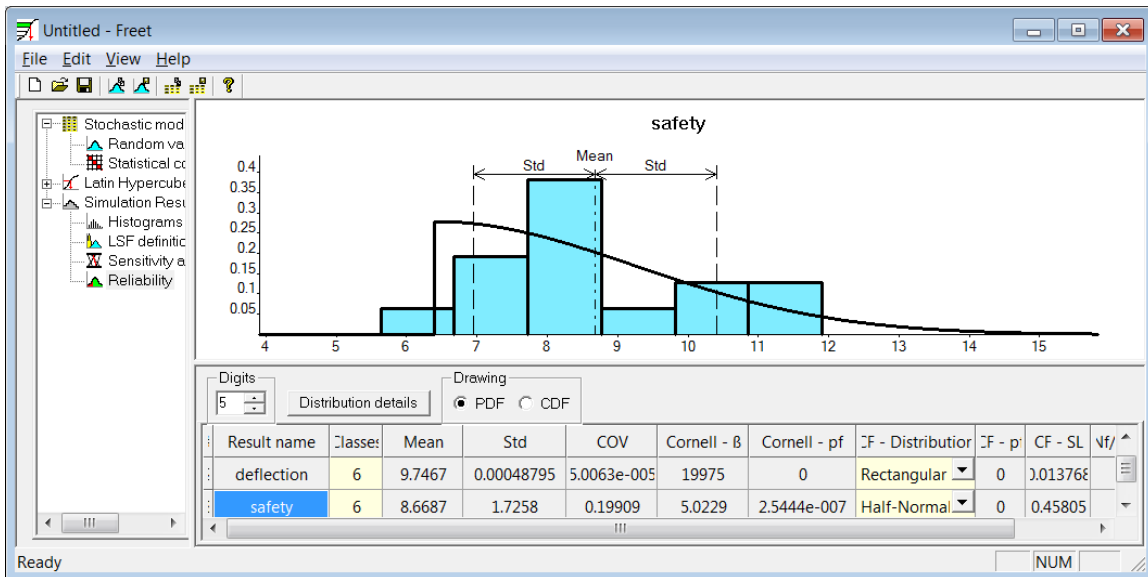
Figure 91 Reliability assessment

4.2.3 Remarks

- Clearly, if a different comparative distribution is selected then a different probability of failure and the reliability index beta is obtained. Figure 92 shows reduced reliability if the 95% quantile of the comparative distribution was kept, but the mean value was 8 instead of 7 and std 0.24 instead of 0.85.
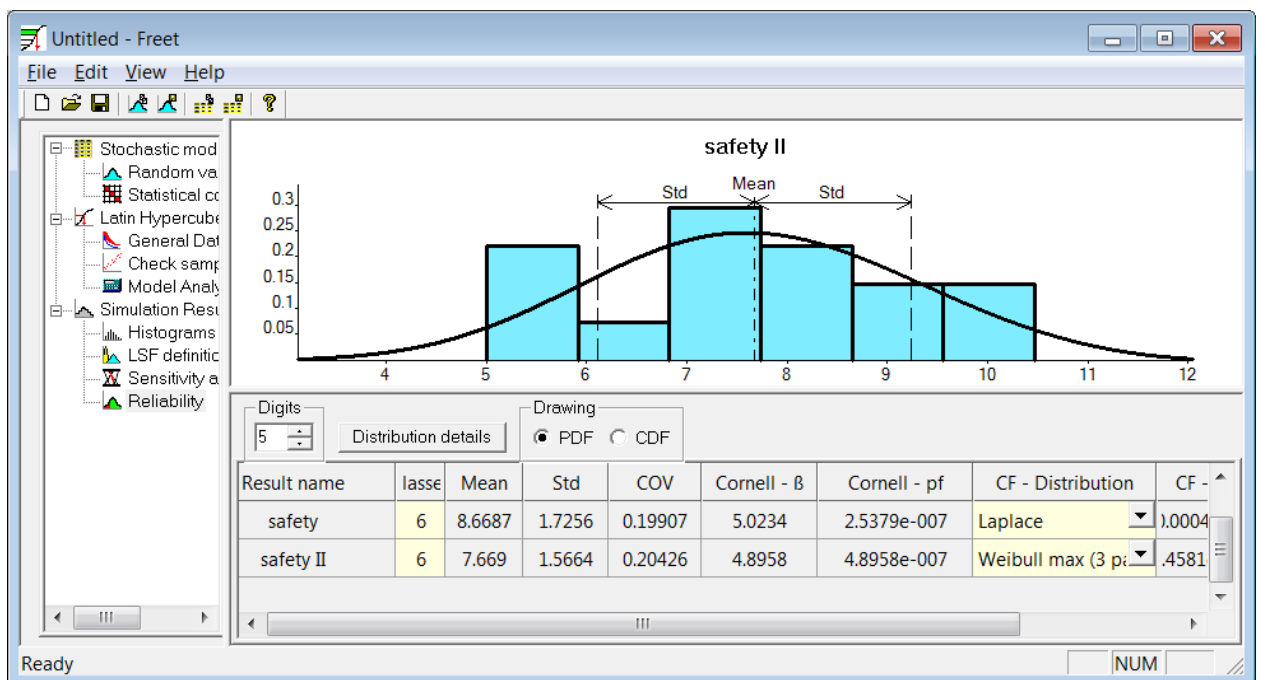

Figure 92 Reliability assessment

- Surprisingly, the highest sensitivity of resistance (i.e. value of monitor **vertical reaction**) is on the tensile strength of concrete ft, followed by fracture energy Gf and compressive strength fc. The least sensitivity of resistance is on the yield stress of rebars.
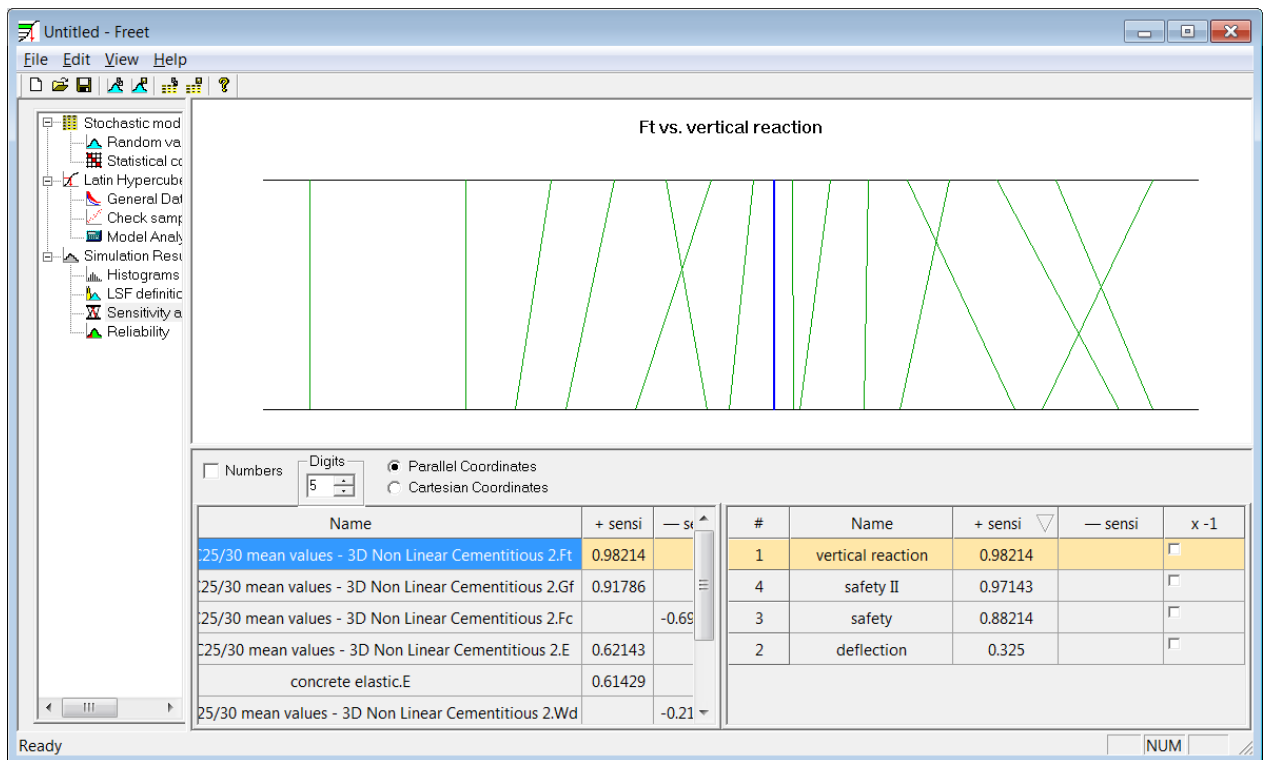
Figure 93 Sensitivity analysis

- If both geometry (modified first example, normal distribution of rebar position with standard deviation 0.01 m) and material properties (this example) are randomized, the safety is reduced and probability of failure raises to 2.15e-6 with the reliability index β = 4,59.

4.3 Serviceability Limit State of a Cantilever

In this example, serviceability limit state is assessed. Computed deflections cover non-linear behaviour of concrete as well as its long-time behaviour due to creep. Probability and reliability index beta is computed when these deflections are compared to two criteria for the SLS: L/500 = 3.2 mm and L/250 = 6.4 mm.

Only variations in material properties are considered. Using EC2, the modulus of elasticity of concrete has been reduced to value E = 8.06 GPa, which corresponds to creep coefficient φ = 2.85. The same value must be used in both Material #1 and #4.

The SLS must be also incorporated into loading cases and analysis steps. In this limit state, only response to quasi-static combination must be assessed. Self-weight of the structure is defined by constant continuous loading 4.0 kN/m (N-R 8 x 0.5 kN/m), while the force acting at the end of the cantilever is reduced to 0.3 multiple of the characteristic load, i.e., 0.6 kN (N-R 1 x 0.6 kN).

```
Material n. 1
 Name : concrete C25/310 mean values LT
 Type: CC3DNonLinCementitious2
 Elastic modulus E = 8.060E+03 [MPa]
 Poisson''s ratio sm = 0.200 [-]
 Tensile strength F_t = 2.600E+00 [MPa]
```

```
Compressive strength F_c = -3.300E+01 [MPa]
Specific fracture energy G_f = 8.368E-05 [MN/m]
Critical compressive displacement Wd = -1.0350E-03 [m]
Eccentricity, defining the shape of the failure surface Exc = 0.520 [-]
Multiplier for the direction of the plastic flow Beta = 0.000 [-]
Specific material weight Rho = 2.300E-02 [MN/m3]
Coefficient of thermal expansion Alpha = 1.200E-05 [1/K]
Fixed smeared crack model will be used Fixed = 1.000 [-]
Plastic strain at compressive strength EPS_CP = -5.000E-04 [-]
Reduction of comp. strength due to cracks fc_LIM = 0.2 [-]
Crack shear stiff. factor s_F = 20.0 [-]
Aggregate size = 0.0200 [m]

Material n. 4
Name : concrete - elastic
Type: CCPlaneStressElastIsotropic
Elastic modulus E = 8.060E+03 [MPa]
Poisson''s ratio sm = 0.200 [-]
Specific material weight Rho = 2.300E-02 [MN/m3]
Coefficient of thermal expansion Alpha = 1.200E-05 [1/K]
```

4.3.1   RLACS project outline:

- Create a new project in **RLACS studio**, select balcony_SLS.cc2 as the basic task.

- Randomize materials similarly to preceding example with one modification: due to high uncertainty of the creep coefficient φ, the C.O.V of the Young's modulus of concrete should be raised, e.g. to 0.25.

- Define correlation, generate randomized samples (e.g. 15) and run analyses.

- The output in **RLACS Studio** should resemble next figure; note that individual LD diagrams are not straight not because of cracking or other non-linear behaviour or concrete or steel, but because of combination of different load cases.
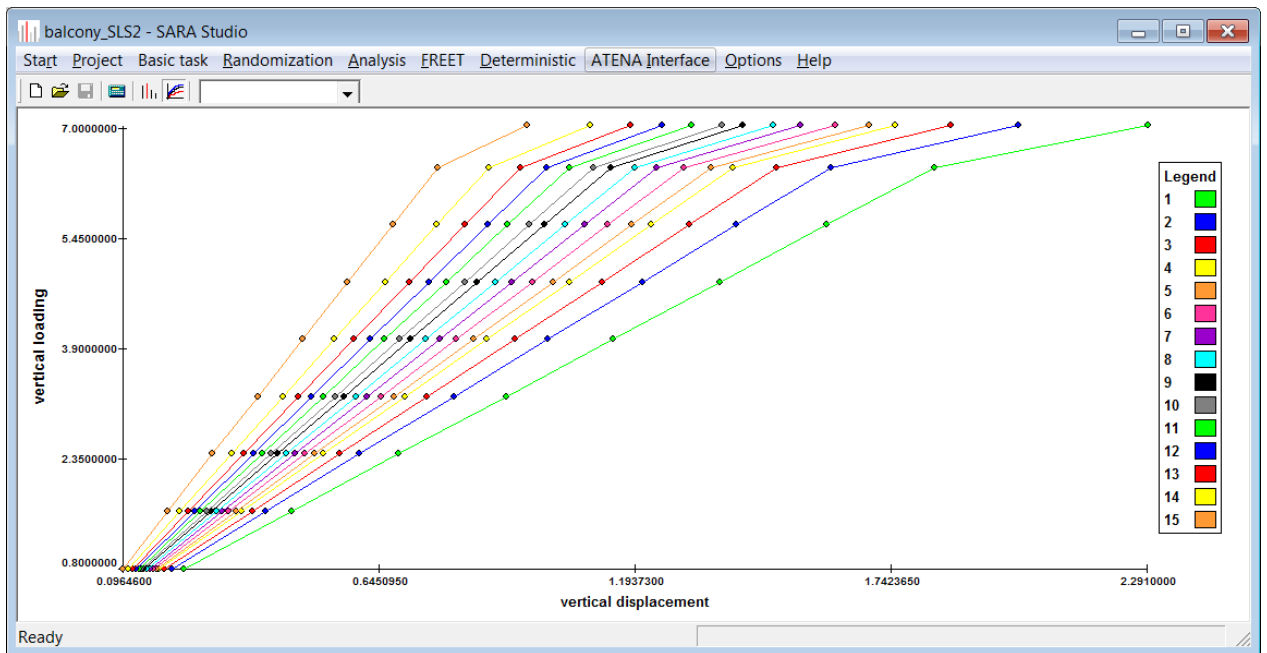

Figure 94 LD diagrams in RLACS Studio.

- Proceed to statistic evaluation in FREET; in tree menu select **Stochastic model | Random variables** and choose tab **Comparative values**. Define two deterministic limit state criteria named *limit deflection 1* and *limit deflection 2*, which correspond to criteria L/500 and L/250 – see Figure 95.
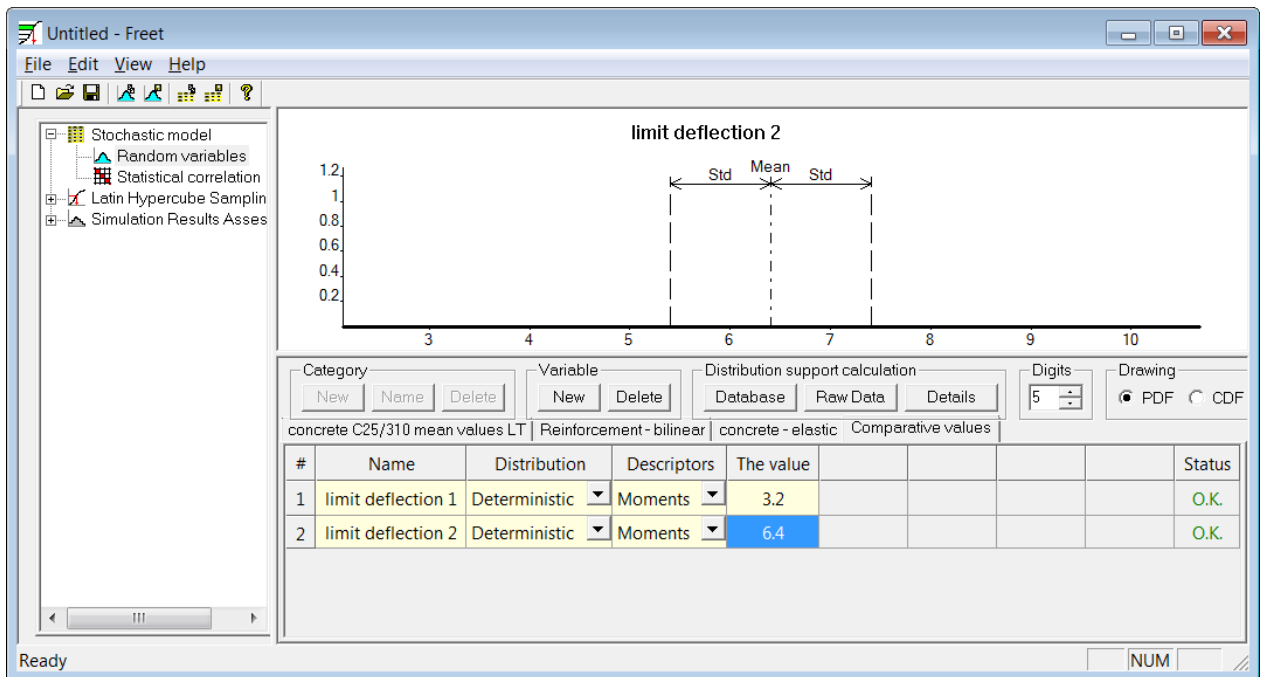

Figure 95 Definition of limit state criteria.

- Use following procedure to sample newly defined limit state functions. Expand **Latin Hypercube Sampling** in the tree menu on the left and select **General Data**. Check **Comparative values only** and then click **Run**.

- In the tree menu expand **Simulation Results Assessment | LSF definition** and define two limit state functions as a difference between recently defined limit criteria *limit deflection 1 (*or *2*) and vertical displacement.
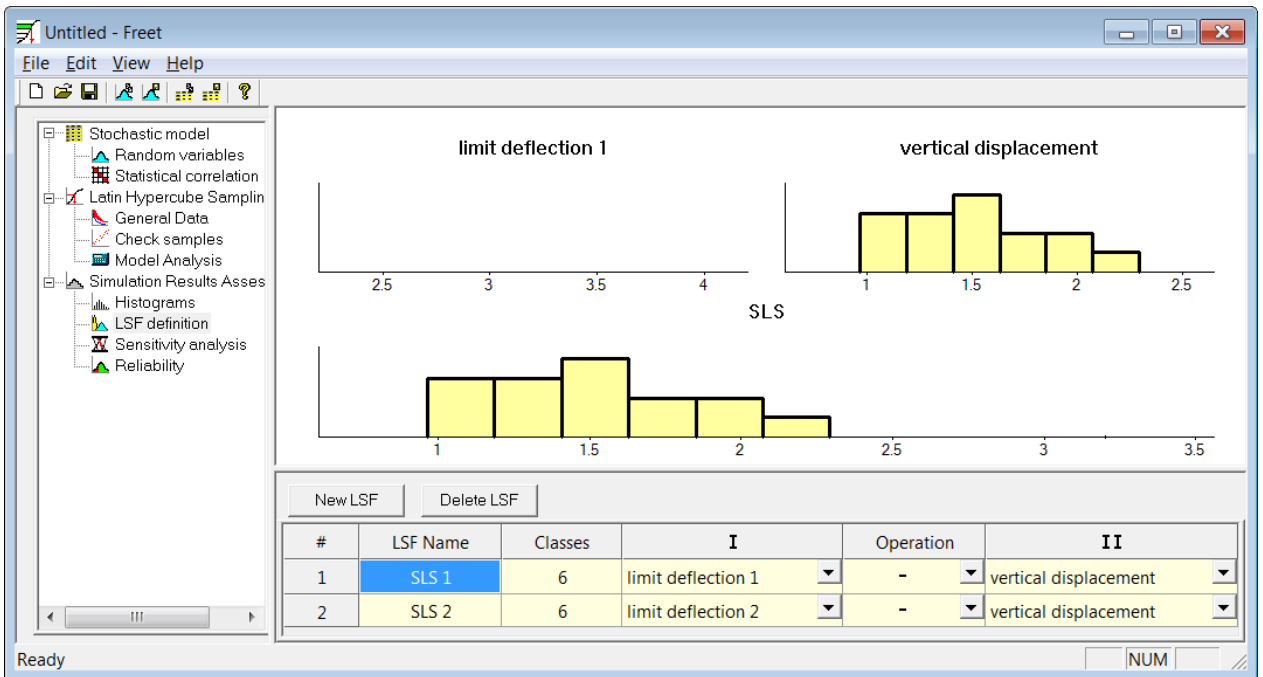
Figure 96 Definition of limit state functions.

Assess reliability. Probability of exceeding stricter deflection criterion (L/500) is 1.33e-5; lenient criterion (L/250) is always fulfilled.
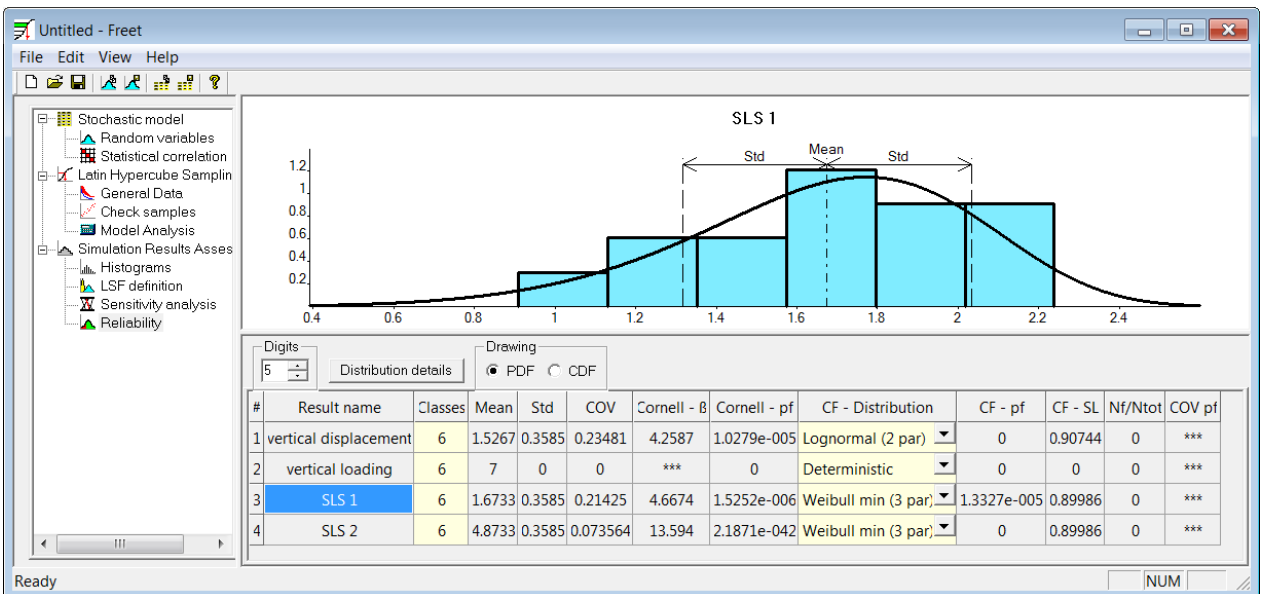

Figure 97 Reliability assessment.

4.3.2 Remarks

- Results of the sensitivity analysis are not surprising: vertical displacement is most dependent on the value of elastic modulus. The sensitivity is in this case -1 for concrete C25/30 and -0.986 for elastic concrete. The reason why these two values differ originates from small error during randomization. Obviously, the correlation between these two variables is not exactly 1.0 as demanded, but something less.
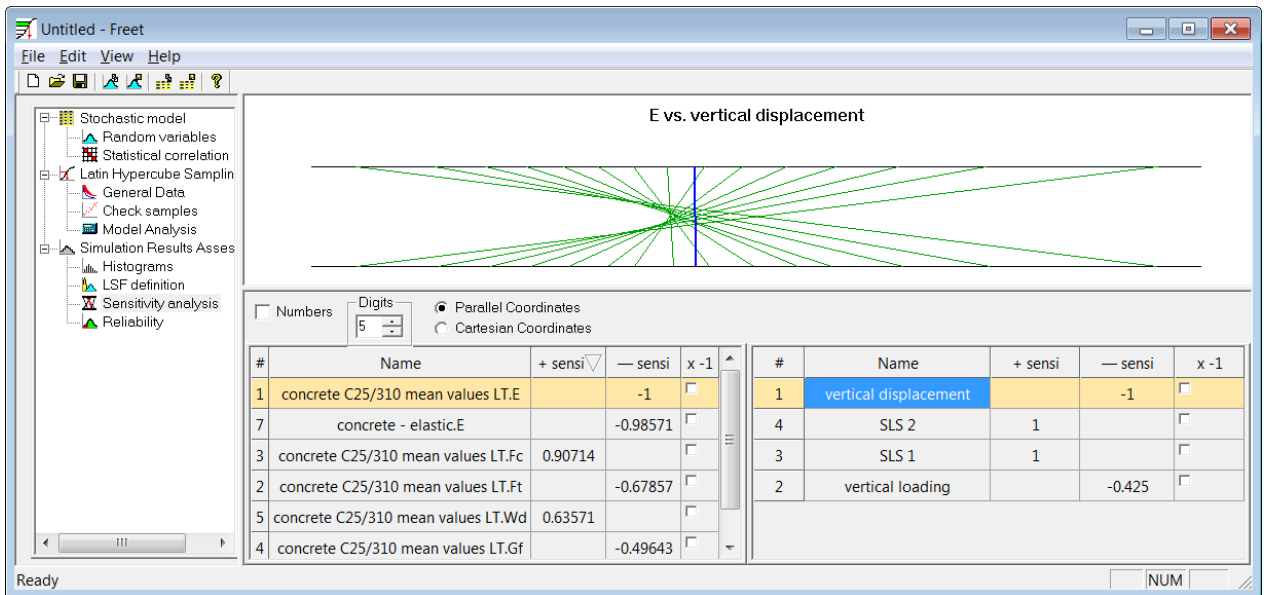
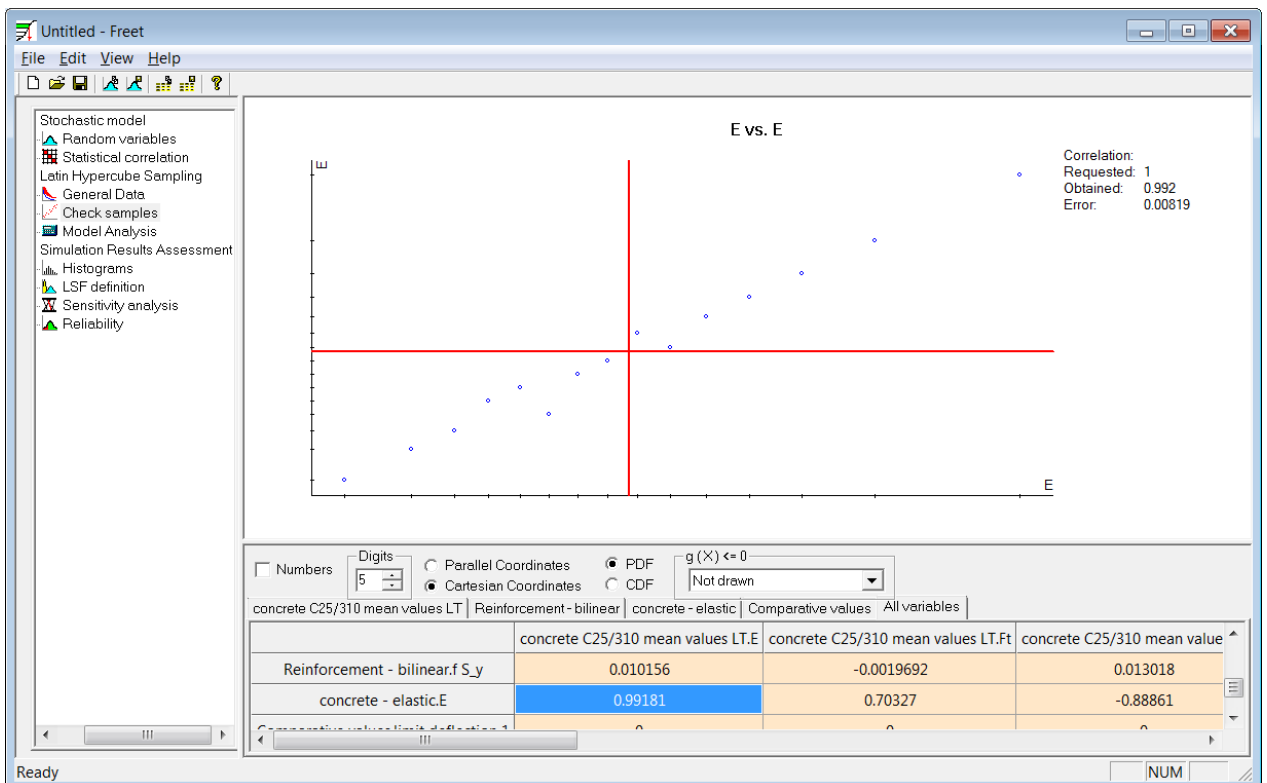**Figure 98 Reliability assessment.**

Figure 99 Obtained correlation between elastic moduli of concrete.

- If the correlation matrix was defined as empty except the diagonal and the perfect correlation between elastic moduli of concrete, then the sensitivity of vertical deflection on remaining material properties would be case zero, because neither tensile nor compressive material strength is exceeded.

- Comparing members of the correlation matrix and the sensitivity of the vertical displacement to see, that these numbers almost coincide (Figure 100).
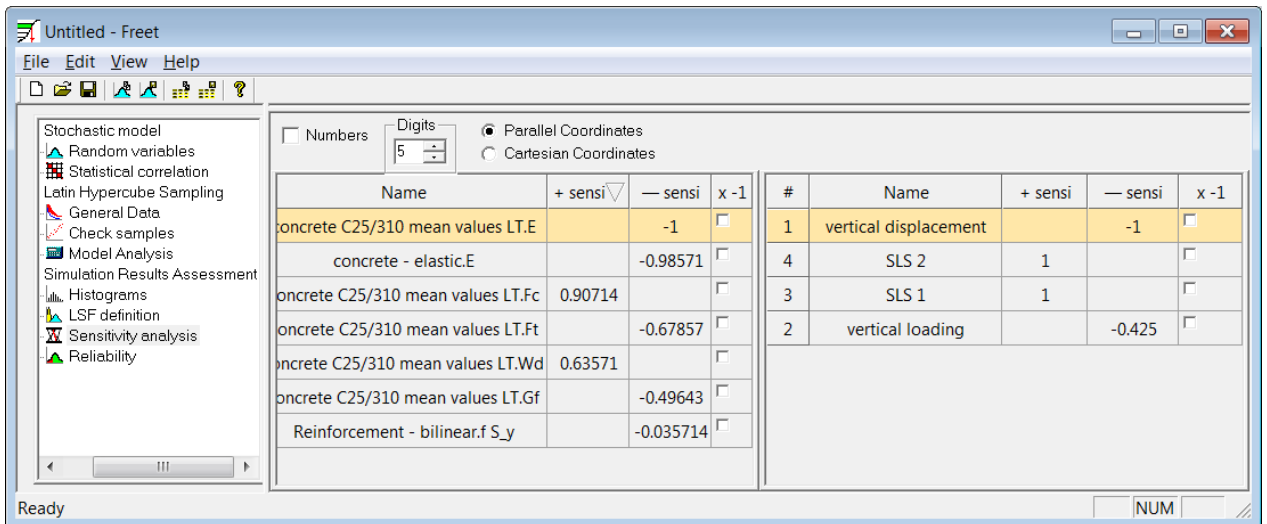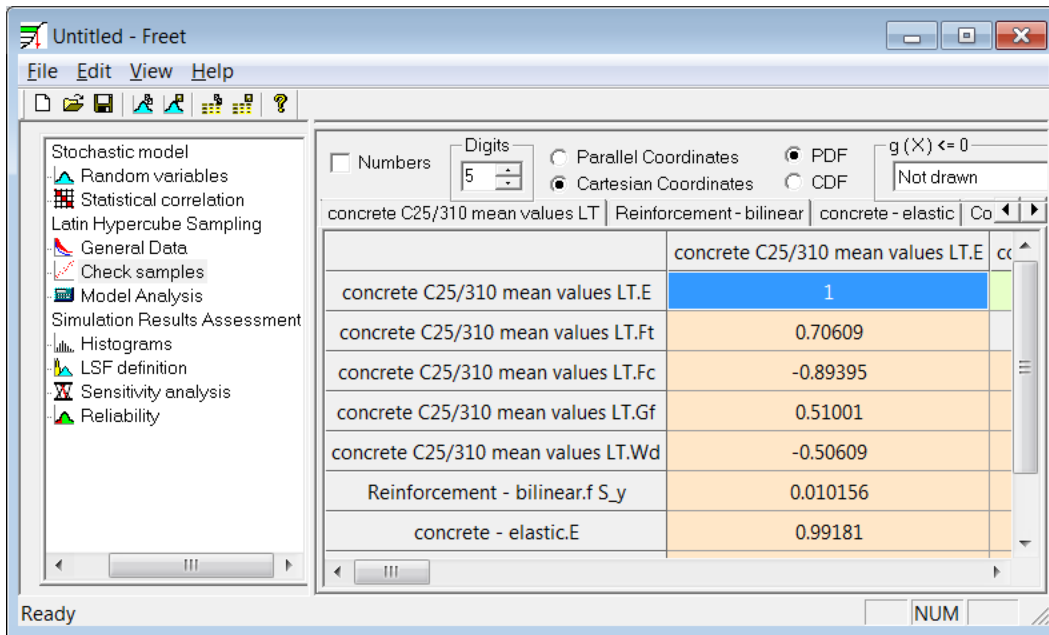
Figure 100 Comparison of quotients from the correlation matrix with sensitivity of vertical displacement.