SIBYL

(Se**I**smic monitoring and vulnera**B**ilit**Y** framework for civi**L** protection)

Agreement number: ECHO/SUB/2014/695550

Deliverable DC3: **Documentation for the developed software tools**

Last modified 15 December 2016

Authors: Prof. Dr. Yury Petryna (TU-Berlin)

Dr. Sergey Tyagunov (TU-Berlin)

Contributors:

Name and Affiliation	Contribution
TU-Berlin	
Yury Petryna Sergey Tyagunov Staffan Langner (student)	Sections 1, 2, 3
Alexandra Bretzke	Technical support

Table of Contents:

1. Introduction

The software tool outlined in this document was developed within the framework of the project SIBYL Action C1: "Simplified Integral Structural Model Approach to Seismic Vulnerability Assessment". Conceptually, as stated in the project proposal, the approach should be, on the one hand, building-specific, implying that every building needs to be individually considered; on the other hand, it should be simple, cost-efficient and fast enough to be applicable to relatively large built-up areas in a comparatively short time (ca. 1 day per building), combining, therefore, a rational assessment rate with an acceptable accuracy of estimation.

The simplified integral structural model (SISM) approach and, correspondingly, the SISM-tool, was developed conforming to the requirements of Eurocode 8[1] and, at the same time, aiming at a simplified evaluation of the seismic performance of existing reinforced concrete buildings using a simplified integral structural model as a system with only a few degrees-of-freedom.

Following the classifications of Eurocode 8, Part 3 (EN 1998-3: 2005), the SISM-Tool was developed bearing in mind the limited knowledge level (KL1) of information required for the seismic evaluation of buildings, assuming that the geometry data is obtained from original outline construction drawings with a sample visual survey or from a full survey, while the details and the material properties are obtained from limited in-situ inspections or/and updated using simulated design. For the seismic evaluation of buildings at this knowledge level, the Eurocode 8 recommends the use of linear analysis methods, either static or dynamic. Correspondingly, the SISM-tool is based on the use of the static approach and the application of a special linearization scheme.

The building assessment procedure generally includes data collection, structural modelling and seismic evaluation. The detailed description of the whole approach is described in Deliverable DC1: "Guidelines for the Building Assessment Procedure and Short-Term Monitoring", while the present Deliverable DC3 describes operating instructions for potential users of the SISM-tool.

With the use of the SISM-tool, the user can assess the seismic performance of a building in a very fast and simple way, entering the collected input information (containing geometry, structural details and material properties of the building and its elements, as well as input parameters of seismic load at the site) and obtaining estimated damage to structural elements, including the damage distribution in the building story-by-story.

The estimated damage level is described in terms of the limit states (LS) traditionally used for reinforced concrete structures: (1) cracking in concrete, (2) yielding in reinforcement bars, and (3) loss of bearing capacity and destruction. It is assumed that, the limit state LS2 approximately corresponds to Damage Limitation (DL) and

[1] http://eurocodes.jrc.ec.europa.eu/showpage.php?id=138

the limit state LS3 – to Near Collapse (NC) according to the classification of performance points in Eurocode 8.

A considerable advantage in applying the offered approach might be the fact that it does not require sophisticated commercial finite element software; instead, the computational procedure is incorporated into Microsoft Excel-based software. There are obvious benefits to this, in particular, Excel is a part of the widely used and commonly available Microsoft Office software package, therefore most engineers and specialists are already familiar with this application. This also implies good opportunities for the training of civil protection practitioners and other potential users of this tool.

The user should bear in mind that the current version of the tool is intended only for reinforced concrete frame buildings. Furthermore, as indicated above, the tool should be used for the knowledge level KL1 in terms of Eurocode 8. The level KL1 (limited knowledge) reflects, in our opinion, the most plausible situations in the routine practice of Civil Protection. If higher knowledge levels are pursued, then more sophisticated methods and software need to be used, which would definitely require additional time and labor resources, as well as specialized qualification and experience of users.

2. Operating Instructions

2.1. The SISM-Tool

The developed SISM-Tool is presented in the form of conventional Microsoft Excel spreadsheets with macros, which were written using the programming language VBA (Visual Basic for Applications). An important prerequisite for using the tool is the activation of macros within Excel, when starting the program for the first time.

Generally, to use the SISM-Tool, the user does not require any special knowledge or experience either in computer programming or in structural mechanics. Using the standard MS Excel interface, the user has to enter collected data (describing characteristics of the investigated building as well as the seismic input for the site) in the properly identified cells of the spreadsheet and then he will automatically obtain the calculated results, including the masses and stiffnesses of the structural elements, the natural frequencies and periods of vibration for different limit states, as well as an evaluation of the expected damage levels and its story-by-story distribution at the given level of seismic hazard for the site.

2.2 The Excel-file and the worksheets

For every new project (i.e., new building to be analyzed), a new Excel-file should be created (using the provided SISM-template) and used for the analysis.

It is important to note that only one Excel-file can be opened and used simultaneously, otherwise uncontrolled interactions between similar tables and cells in different Excel-files is possible, which may lead to erroneous results in the calculations.

Initially, the template-file contains two worksheets intended directly for the user. The first worksheet is titled "Building", where the required input parameters (see a detailed description in Table 3.1 of this document) are to be entered by the user and where the output results of the building’s evaluation are presented. The last sheet of the file (entitled “Manual”) contains these operating instructions.

In the process of entering the required input data, additional worksheets (up to eighteen in total) are generated automatically, depending on the entered number of stories (up to ten in the current version of the tool) in the building and the number of types of columns (up to five different cross-sections) in the structural system. One more worksheet is purposed for the case when a stiffness core (shaft) is present in the investigated building. The additionally generated worksheets are used by the computational program and contain intermediate results of computations and temporary data for individual stories of the building. Generally, those worksheets are not intended for normal users; however, they can be used for reviewing and controlling intermediate results by more qualified users.

For a description of the main parameters used in the tool, the user should refer to Table 3.1 in Section 3 of this deliverable.

2.3 Worksheet “Building”

The main worksheet of the tool, titled "Building", is intended for the user. The worksheet contains both the tables for entering the input data and the output results of the computations. The required input data includes parameters of the building and its structural elements, as well as the parameters of the seismic loading at the site. The output data includes calculated masses and stiffnesses, natural frequencies of the building as well as evaluation of its seismic performance, including expected damage (in terms of achieved Limit States for different stories of the building).

2.3.1 Parameters of the building

The first table in which the user should enter the parameters of the building is located in the upper part of the worksheet (Fig.1). Initially, the cells into which the user has to enter the required parameters are marked in pink (Fig.1a) and they automatically turn light green (Fig.1b) as soon as the corresponding data have been entered. In general, the presence of pink cells in the worksheet is intended to prompt the user to enter specified data in the corresponding cells.

Building

Number of Stories		Length X [m]		E-Modulus [MN/m²]		Poisson's ratio		E-Modulus, Walls [MN/m²]	Meas./Calc. Frequency, X [Hz]			0
Total Height		Length Y [m]		Siffness Ratio		Number of Column Types			Meas./Calc. Frequency, Y [Hz]			0

(a)

Building

Number of Stories	5	Length X [m]	33.6	E-Modulus [MN/m²]	33000	Poisson's ratio	0.2	E-Modulus, Walls [MN/m²]	Meas./Calc. Frequency, X [Hz]	1.72		1.601816
Total Height	29.5	Length Y [m]	25.5	Siffness Ratio	2	Number of Column Types	1	5000	Meas./Calc. Frequency, Y [Hz]	1.6		1.7673342

(b)

Building

Number of Stories	5	Length X [m]	33.6	E-Modulus [MN/m²]	33000	Poisson's ratio	0.2	E-Modulus, Walls [MN/m²]	Meas./Calc. Frequency, X [Hz]	1.3		1.601816
Total Height	29.5	Length Y [m]	25.5	Siffness Ratio	2	Number of Column Types	1	5000	Meas./Calc. Frequency, Y [Hz]	1.6		1.7673342

(c)

Figure 1: Table with parameters of the building (a - initial view without data, b – an example of the table with entered data and acceptable model, c – an example, when the model has to be corrected)

The building parameters (as can be seen in Fig.1) include the total number of stories, total height of the building and its overall dimensions in plan (in both X and Y directions). In the same table, one should also enter the number of column types in the building (i.e., the number of different cross-sections, which in the current version of the tool is limited to five) and specify the parameter of "stiffness ratio" between horizontal and vertical structural elements (beams and columns) of the building. In addition, the user should enter the mechanical parameters of the structural concrete (E-Modulus and Poisson's ratio) and E-Modulus of the walls. The user should use the measure units as indicated in the tables.

Additionally, the measured (or estimated) natural frequencies for the first two bending modes in both directions should be entered. The last column of the table contains the automatically calculated values of the natural frequencies, and compared with the measured frequencies. The results of such comparison are shown in the last, but one column using the traffic light indicator, namely, the results shown in green color (Fig.1, b, when the calculated differences between the measured and numerical values lie within 20%) are acceptable for the further analysis, while in case of a red color (Fig.1, c, when the differences are larger than 20%) the parameters of the model should be corrected to achieve a better fit. In the latter case, improvement in the model should be done by changing those building characteristics that are associated with the largest uncertainties in the available input data (e.g., they may be related to unknown cross-section sizes or the mechanical properties of the building components).

2.3.2 Columns, Shafts and Walls

The next four tables of the worksheet "Building" (namely, "Uniform Grid of Columns", "Non-uniform Grid of Columns", "Hollow Shaft", "Walls") are intended for entering the parameters of the main structural elements (Fig. 2). The data should be entered in the corresponding table rows (story by story). Having entered the data for one story, the user should press the button "Insert" at the end of the table to transfer the data for the current story and after that, when the cells in the table row are cleared, the data can be entered for the next story and subsequently transferred by pressing the button "Insert" for every story.

Uniform Grid of Columns

Current Story	Total number of columns in X-direction	Total number of columns in Y-direction	Bay Width, X [m]	Bay Width, Y [m]	Cross-section Width, X [m]	Cross-section Width, Y [m]	Diameter [m] (if round cross-section, else 0)	First Column Coordinate, X [m]	First Column Coordinate, Y [m]	Material Density [kN/m3]	INSERT

Non-uniform Grid of Colums

Current Story	Column Coordinate, X [m]	Column Coordinate, Y [m]	Cross-section Width, X [m]	Cross-section Width, Y [m]	Diameter [m] (if round cross-section, else 0)	Material Density [kN/m3]	INSERT

Hollow Shaft

Current Story	X-coordinate (center) [m]	Y-coordinate (center) [m]	Cross-section Width, X [m]	Cross-section Width, Y [m]	Wall Thickness, X [m]	Wall Thickness, Y [m]	Material Density [kN/m3]	INSERT

Walls

Current Story	First Point Coordinate in Plan, X1 [m]	First Point Coordinate in Plan, Y1 [m]	End Point Coordinate in Plan, X1 [m]	End Point Coordinate in Plan, Y1 [m]	Wall Thickness, [m]	Material Density [kN/m3]	INSERT

Figure 2: Tables with the parameters of the main structural elements

2.3.2.1 Uniform Grid of Columns

This table should be used in the case where the structural system is represented by a uniform grid of columns. The data to be entered are: total number of columns and bay widths in both the X and Y directions, cross-section sizes (for rectangular columns) or diameter (for round cross-sections), coordinates of the starting point of the grid and material density (Fig. 2). As above, the user should enter and transfer the required data story by story and pressing the button “Insert” at the end of the table row for each story.

2.3.2.2 Non-uniform Grid of Columns

This table should be used for the case of a non-uniform grid of columns. The data to be entered for the columns are: coordinates of the columns, cross-section sizes (for rectangular columns) or diameter (for round cross-sections), and the material density (Fig. 2). Again, the user should enter and transfer the required data story by story and pressing the button “Insert” at the end of the table row for each story.

2.3.2.3 Hollow Shaft (Stiffness Core)

This table should be used for cases involving the presence of a stiffness core (e.g., an elevator shaft) in the building, which can significantly contribute to its lateral-load-resisting capacity. The data to be entered are: the coordinates of the center of the core and cross-section sizes, including of the widths of the core and the thicknesses of the core walls in the both directions, as well as the material density (Fig. 2). In the same way as for the tables above, at the end of the table row the user should press the “Insert” button for each story.

2.3.2.4 Walls

This table should be used if walls are present in one or another story and they are to be considered in the simplified model (in case they contribute to the load-resisting-capacity or to the stiffness of the structural system). The required parameters are: the coordinates of the starting point of the wall and the end point of the wall, thickness of the wall and the material density (Fig. 2).As before, having entered the required data for a current story, the user should press the “Insert” button at the end of the table row to transfer the data.

Important note: The building parameters entered in these four tables (uniform and non-uniform grids of columns, hollow shaft, and walls) do not appear in the tables in the worksheet “Building”. Instead, corresponding tables with the entered parameters are created in additionally generated worksheets (for corresponding stories of the building). As mentioned above, the additional worksheets can be used for reviewing and controlling the intermediate results by more qualified users. For other users, if some of the parameters entered in the four tables above need to be modified, it is recommended to use for the modified model a new file of the SISM-template.

2.3.3 Number of Elements

The next table of the worksheet is entitled “Number of Elements” and contains information about the total number of columns, shafts and walls for every story (Fig. 3). The table is useful for checking if all the required data have been entered properly for each story. The size of the table (i.e., the number of rows) is defined by the value entered in the cell "Number of Stories" at the beginning of the worksheet (Fig. 1). As for the general building parameters, the cells in the table are initially pink and they turn light green when a value is entered. If some of the elements (columns, shafts or walls) are not present in a particular story, then a zero value should be entered in the corresponding cells.

Number of Elements

Story	Number of columns	Number of shafts	Number of walls
1	32	1	2
2	32	1	8
3	32	1	8
4	32	1	8
5	32	1	8

Figure 3: Table for controlling the number of structural elements for each story.

This table is intended for controlling whether all the columns, walls, or shafts have been included properly, therefore it would be an advantage to fill the table before entering the data for the structural elements. For example, if not all columns have been entered yet, the cells in the tables "Uniform Grid of Columns" and "Non-uniform grid of columns" are pink. This indicates that the user should enter the data for at least one column. At the same time, if one enters a current story number in "Current Story" cell (Fig. 2), the cells turn green again, if all the columns have been already entered in this story. Thus, it means that the missing data may refer to another story. An advanced user (i.e., more qualified in the field of structural mechanics) may have an overview of all the entered data in the additional worksheets related the corresponding stories.

2.3.4 Cross-sections of columns and shaft and concrete properties

The next tables are intended for entering parameters of the main structural elements (the columns and the shaft) to be used for the analysis of their bearing capacity, in particular, corresponding to achieving different limit states in the elements.

In the table "Column cross-sections", the geometry data should be entered for the different column types (Fig. 4). The size of the table (number of rows) is defined correspondingly, depending on the number of column types with different cross-sections. For every column type, the user should enter data about the cross-section sizes and the concrete cover (Fig. 4). Up to ten different column profiles (cross-sections) can be considered in the current version of the tool.

Column cross-sections

Current No.	X-Width [m]	Y-Width [m]	Radius [m]	Concrete cover [m]
1	0.5	0.75	0	0.03

Figure 4: Geometry data for different column types.

As for the shaft (representing the stiffness core), the current version of the tool considers the possible presence of only one shaft in the building. The geometry data for the shaft are to be entered in the table "Shaft cross-sections" (Fig. 5).

Shaft cross-sections

Current No.	X-Width [m]	Y-Width [m]	Wall Thickness, X [m]	Wall Thickness, Y [m]
1	1.7	3.9	0.15	0.15

Figure 5: Geometry data for the shaft.

The entered data are then compared with the previously entered data for different stories (Fig. 2) and used for the analyses of the structural performance of the elements for different limit states. The data provided in the table “Concrete properties” serves the same purpose, where the user has to enter the strength values and the reinforcement ratio (Fig. 6). If the required input parameters are not available from the in-situ testing or from the building specifications, then, as a first approximation, the user can use default values for the time of the construction of the building.

Concrete Properties	
Strength [MPa]	30
Reinforcement ratio	0.015
Tens. strength [MPa]	2.2

Figure 6: An example of concrete properties

2.3.5 Stiffnesses, Masses and Weights

The next two tables contain the calculated data about the story-by-story distribution of stiffnesses, masses and weights in the building (Fig. 7). Initially, the tables are empty, but, when the parameter “Number of Stories” has been entered (Fig. 1), these tables are extended correspondingly, in the same way as the table “Number of Elements” (Fig. 3). The values of “Story Height”, “Slab Thickness” and “Slab density” are the only parameters to be entered in the table “Masses and Weights” (Fig. 7, b) by the user; all other fields are automatically filled out with the calculated values based on the previously entered input data. The slab density for concrete is usually equal to 25 kN/m^3. It is, however, recommended to increase this value by 20% to 50% to take into account the weight of the partition walls and non-structural elements.

Stiffnesses [MN/m]

Story	Kx	Ky
1	1224.574208	1867.018663
2	7560.66339	4861.827031
3	7560.66339	4861.827031
4	7560.66339	4861.827031
5	9420.398476	6396.379231

(a)

Masses and Weights					
Story	Story Mass [kN/m]	Slab density [kN/m^3]	Story Height [m]	Slab Thickness [m]	Weight [kN]
1	364.95	50	5.8	0.4	19961.415
2	783	50	5.1	0.4	20816.1
3	783	50	5.1	0.4	20816.1
4	783	50	5.1	0.4	20385.45
5	783	50	4	0.4	18545.4

(b)

Figure 7: The tables with distribution of stiffnesses (a) and masses and weights (b).

2.3.6 Stiffnesses for different limit states

The following tables contain the values of stiffnesses calculated for the different limit states (LS2 and LS3), taking into consideration the changed physical and mechanical characteristics of the structural elements in accordance with the achieved damage level (limit state). The changed parameters are calculated based on the previously entered input information and, subsequently, the tables are filled out automatically (Fig. 8).

Stiffnesses for LS2 (DL) [MN/m]		
Story	Kx	Ky
1	305.1640621	651.6933565
2	9184.140389	5061.715481
3	9138.768804	4973.164782
4	9088.627644	4871.305466
5	11637.21649	6903.160993

(a)

Stiffnesses for LS3 (NC) [MN/m]		
Story	Kx	Ky
1	86.58091971	269.8508604
2	6428.235962	3404.366715
3	6400.871616	3317.019164
4	6380.100021	3248.437595
5	8226.057853	4738.445971

(b)

Figure 8: Calculated stiffnesses for limit states LS2 (a) and LS3 (b).

2.3.7 Earthquake forces corresponding to different limit states

For the different limit states, when the building properties have changed due to the damage experienced under seismic loading, the distribution of forces in the damaged building is changed correspondingly. The following tables show the distribution of earthquake forces in the building story-by-story and in both the X and Y directions. The computed values correspond to different stiffnesses under different limit states: LS1 (a), LS2 (b) and LS3 (c).

Earthquake Forces for LS1 [MN/m]

Story	Force x	Force y
1	44.06530566	45.38263447
2	49.38422486	55.89355618
3	51.8841324	62.44059179
4	52.46205296	65.5300738
5	48.32252537	61.07221202

(a)

Earthquake Forces for LS2 (DL) [MN/m]

Story	Force x	Force y
1	17.98147721	26.8954629
2	19.32044418	31.14174844
3	19.70375037	33.4901457
4	19.54894949	34.38935262
5	17.87441826	31.79987759

(b)

Earthquake Forces for LS3 (NC) [MN/m]

Story	Force x	Force y
1	5.658367865	14.67669673
2	5.98487053	16.35289366
3	6.033254285	17.13198717
4	5.940190266	17.30271417
5	5.415223879	15.90541251

(c)

Figure 9: Distribution of earthquake forces for LS1 (a), LS2 (b), LS3 (c)

2.3.8 Calculated frequencies and periods of vibration

The next table (Fig. 10) contains the calculated frequencies for bending vibrations in the X and Y directions, corresponding to different limit states. Only the values for the first vibration modes are considered here, because only the first bending modes are of practical interest in the framework of the simplified approach.

As mentioned above, the calculated frequencies for the initial state of the building are compared with the measured values (Fig. 1) to validate and, if necessary, to calibrate the created structural model of the building. Furthermore, the obtained values (recalculated to vibration periods, Fig. 10) are used for determining the earthquake forces on the structures (Fig.11).

Calculated Frequencies [Hz]			Periods [s]	
Limit State	X	Y	X	Y
1	1.658379002	1.834140218	0.602998469	0.545214586
2	0.85992011	1.18960372	1.162898725	0.840616067
3	0.463357946	0.787138196	2.158158738	1.270424946

Figure 10: Tables with calculated frequencies and periods of vibration.

2.3.9 Earthquake base shear force

Another table, where the user has to enter the input data is the table "Base shear force" (Fig. 11).

In accordance with Eurocode 8, for the calculation of the earthquake base shear force F_b the following expression is used:

$$F_b = S(T_1) * m * \lambda ,$$

where $S(T_1)$ is the ordinate of the acceleration spectrum at the fundamental period of vibration of the building for lateral motion in the considered direction ($T_1 = 1 / f_1$). The user has to determine this value, either using available results of seismic hazard assessment for the site (e.g., seismic microzonation) or following the national seismic code and taking into consideration the estimated natural frequencies (and vibration periods) for the building (Fig. 10); the user has to enter the corresponding values of spectral acceleration for the vibration periods estimated for the three limit states (Fig.10).

m is the total mass of the building, above the foundation or above the top of a rigid basement. In the SISM-tool, this value is determined automatically, using the previously calculated results (Fig. 7, b).

λ is the correction factor, the value of which is equal to: $\lambda = 0{,}85$ if $T_1 < 2\ T_C$ and the building has more than two stories, or otherwise $\lambda = 1{,}0$ (EN 1998-1: 2004).

When the parameters listed above are entered, then the maximum earthquake force (base shear force) F_{tot} is calculated automatically (Fig. 11) and used for further analyses, including the calculation of earthquake force distributions (Fig. 9) as well as for the seismic evaluation of the building (Fig. 12).

Base Shear Force

Lambda LS1 x	1	Spectral acceleration LS1 x	2.45	Shear Force F_{tot} LS1 x [MN]	246.1182413
Lambda LS1 y	1	Spectral acceleration LS1 y	2.89	Shear Force F_{tot} LS1 y [MN]	290.3190683
Lambda LS2 x	1	Spectral acceleration LS2 x	0.94	Shear Force F_{tot} LS2 x [MN]	94.4290395
Lambda LS2 y	1	Spectral acceleration LS2 y	1.57	Shear Force F_{tot} LS2 y [MN]	157.7165873
Lambda LS3 x	0.85	Spectral acceleration LS3 x	0.34	Shear Force F_{tot} LS3 x [MN]	29.03190683
Lambda LS3 y	1	Spectral acceleration LS3 y	0.81	Shear Force F_{tot} LS3 y [MN]	81.36970425
Weight [MN]	100.456425				

Figure 11: Calculation of base shear force.

2.3.10 Limit State Assessment

Finally, the tables „Limit State Assessment“ represent the results of the building assessment (Fig. 12), where the estimated values of forces corresponding to occurrence of different limit states (structural capacity) are compared with the actual earthquake forces corresponding to the hazard assessment for the site (structural demand). The comparison is made separately for the X and Y directions.

The results of comparison are presented in the table using conventional “traffic light colors”, that is green, when the earthquake loads are lower than the forces corresponding to a certain limit state, and red, if the actual loads are higher than the corresponding forces for the limit states.

The presented results can be interpreted in the following simple way: the green color shows that the corresponding limit state (LS1, LS2 or LS3) is not achieved at that story, while red shows that the limit state is achieved at the given story under the given level of seismic load.

Correspondingly, the appearance of red color for LS2 (corresponding to Damage Limitation) and, especially for LS3 (corresponding to Near Collapse) for one of the stories of the building under consideration shows that the building is not safe at the given level of hazard and, therefore, protection measures should be taken for retrofitting/strengthening of the structural system.

Limit state assessment

X-direction

Story	EQ Force LS1 [MN]	EQ Force LS2 [MN]	EQ Force LS3 [MN]	LS1 Force [MN]	LS2 Force [MN]	LS3 Force [MN]
1	22.4682336	14.5382688	5.782266	1.815516867	11.81378224	21.84693608
2	20.62119746	13.48356125	5.368740108	1.462243797	10.42359633	22.5082333
3	17.07766431	11.35841487	4.522249012	1.10897072	9.014034361	23.71211442
4	12.12617218	8.228690871	3.267346059	0.755697696	7.5809278	25.60760142
5	6.169199464	4.266421568	1.684963293	0.402424539	6.117250942	28.32824942

Y-direction

Story	EQ Force LS1 [MN]	EQ Force LS2 [MN]	EQ Force LS3 [MN]	LS1 Force [MN]	LS2 Force [MN]	LS3 Force [MN]
1	21.8074032	15.1990992	7.2691344	2.16578976	13.82014828	28.02515187
2	20.01469165	14.05817093	6.7297563	1.744358701	12.27850647	29.56361822
3	16.57538007	11.78955111	5.6433911	1.322927632	10.71774427	31.90759017
4	11.76952006	8.495305397	4.057826844	0.901496627	9.132985772	35.27303927
5	5.987752421	4.381763008	2.084418946	0.480065463	7.515586419	39.8529442

Figure 12: Comparison of earthquake forces and limit state assessment

2.4 Worksheet „Manual“

The last sheet of the Excel-file (entitled “Manual”) contains these operating instructions.

3. Description of parameters used in computations

Table 3.1 provides a synoptic description of the main input and output parameters, including the corresponding measure units. This description is given for the users who are not familiar with the terminology specific for the practice of structural and earthquake engineering.

Table 3.1: Description of used parameters

Parameter	Description
Number of Stories	Number of stories above the ground
Total Height	Total height of the building above the ground, m
Length (X, Y)	Overall dimensions of the building in plan in X and Y directions, m
E-modulus	Module of elasticity of concrete, MN/m^2
Poisson's Ratio	The Poisson's ratio value for concrete
Stiffness Ratio	Stiffness ratio between horizontal and vertical structural elements (beams and columns) of the building
Number of Column Types	The number of column types with different cross-sections (either shape or sizes)
E-Modulus Walls	Module of elasticity of walls
Measured Frequency (X,Y)	The values of bending eigenfrequencies of the building obtained from in-situ measurements for two directions (X,Y), Hz
Current Story	The story for which the parameters are currently entered
Total Number of Columns in X, Y direction	The total number of columns in both directions in plan in case of uniform grid of columns
Bay Width (X, Y)	Distance between columns in both directions in case of uniform grid of columns, m

Cross-section Width (X,Y)	Sizes of the cross-sections of the columns, m
Diameter	Diameter of the round cross-section, m
First Column Coordinate (X,Y)	Coordinates of the first columns in case uniform grid of columns, m
Material Density	Density of the material (for structural concrete or masonry walls, correspondingly), kN/m^3
First Point Coordinate (X,Y)	X and Y coordinates of the point where the wall starts, m
End Point Coordinate (X,Y)	X and Y coordinates of the point, where the wall ends, m
Strength	Compression strength of concrete, MPa
Reinforcement Ratio	Ratio of the effective area of the reinforcement to the effective area of the concrete in the cross-section of a structural member.
Tens.strength	Tensile strength of reinforcement steel, MPa
Earthquake forces	Earthquake forces at different levels (center masses corresponding to the building floors)
Base shear force	Maximum earthquake force at the base of the building
Limit State 1 (LS1)	Fist limit state of reinforced concrete elements, corresponding to occurrence of cracks in concrete
Limit State 2 (LS2)	Second limit state of reinforced concrete elements corresponding to yielding in reinforcement steel bars
Limit State 3 (LS3)	Third limit state of reinforced concrete corresponding to the structural failure