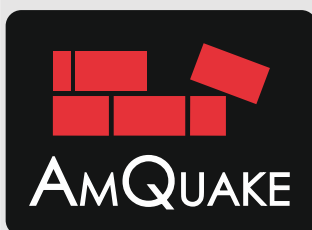




Cervenka Consulting Ltd.
Na Hřebenkách 55
150 00 Prague
Czech Republic
Phone: +420 220 610 018
E-mail: cervenka@cervenka.cz
Web: <http://www.cervenka.cz>



AMQUAKE

Program Documentation



Written by
Jan Červenka
Libor Jendele
Zdeněk Janda

Prague, April 12, 2016

Trademarks:

AmQuake is registered trademark of Cervenka Consulting, Ltd.

Microsoft and Microsoft Windows are registered trademarks of Microsoft Corporation.

Other names may be trademarks of their respective owners.

Copyright © 2010-2016 Cervenka Consulting Ltd.

CONTENTS

LIST OF FIGURES	V
LIST OF TABLES	VIII
1 INTRODUCTION	1
2 INSTALLATION AND REGISTRATION	2
2.1 Installation process	2
2.2 Registration process	4
2.3 Registration process for student (university) version	9
3 USER'S MANUAL	10
3.1 Main window	11
3.2 Main menu bar	12
3.3 General data	15
3.3.1 Analysis information	15
3.3.2 Analysis parameters	15
3.3.3 Seismic parameters	22
3.4 Materials	26
3.4.1 Wall materials	28
3.5 Topology	41
3.5.1 Rules for DXF files	43
3.5.2 Import DXF	47
3.5.3 Topology – walls / columns	52
3.5.4 Topology – windows / doors	53
3.5.5 Topology – ceilings	54
Loading	57
3.5.6 Ceiling load	57
3.6 Mesh	58
3.6.1 Nodes	61
3.6.2 Elements	62
3.6.3 Rigid links	63
3.7 Calculate	64
3.8 Results	68
3.9 Analysis	68
3.10 Print statement	79
4 HOW TO DO	80

4.1	How to model columns	80
4.2	How to model confined masonry	80
4.3	How to model reinforced masonry	84
4.4	How to model infill masonry	85
4.5	How to model reinforced concrete elements	86
4.6	How to improve convergence and reduce errors	86
4.7	How to model r.c. ring	87
5	THEORY MANUAL	90
5.1	Pushover Analysis	90
5.1.1	<i>EN 1998-1 assumptions for pushover analysis</i>	92
5.1.2	<i>Target displacement</i>	94
5.2	Equivalent frame method	99
5.2.1	<i>Finite element model</i>	99
5.2.2	<i>Rigid body connections</i>	100
5.2.3	<i>Connection between walls in adjacent floors</i>	103
5.3	Nonlinear behaviour	105
5.3.1	<i>Masonry</i>	105
5.3.2	<i>Reinforced concrete</i>	110
5.3.3	<i>Element strength evaluation</i>	113
6	CONCLUSIONS	120
7	LITERATURE	121
8	APPENDIX A	122

LIST OF FIGURES

Figure 1: Selection of national language during the installation process.....	3
Figure 2: Selection of program components to be installed.....	3
Figure 3: The About dialog with the licence activation button.....	5
Figure 4: License activation dialog	6
Figure 5: AmQuake online license activation, step 1/3	7
Figure 6: AmQuake online license activation, step 2/3	7
Figure 7: AmQuake online license activation, step 3/3	8
Figure 8: Final step in the program activation.....	8
Figure 9: Final step in the program activation – student version.....	9
Figure 10: AmQuake screen and description of model zoom and rotation using the mouse buttons.....	11
Figure 11: Main menu bar	12
Figure 12: File menu items	12
Figure 13: Selection of the country specific standard from the menu File New file.	13
Figure 14: Settings menu items.....	13
Figure 15: Selection of simple and standard user mode.....	14
Figure 16: Help menu	14
Figure 17: Analysis information	15
Figure 18: Analysis parameters – Building assessment standard.....	16
Figure 19: Seismic parameters	22
Figure 20: Ground types classification table (NOTE: this table may be specified for each country in its national annex)	23
Figure 21: Typical shape of the elastic response spectra, (see Section 5.1).....	25
Figure 22: Material definition	27
Figure 23: Catalogue of materials based on Wienerberger products, reinforcement and concrete types	27
Figure 24: Unreinforced masonry wall material.....	28
Figure 25: Stress-strain relationship of masonry.....	29
Figure 26: Dialog for reinforced masonry – standard sheet	30
Figure 27: Dialog for reinforced masonry – vertical reinforcement definition.....	31
Figure 28: Dialog for reinforced masonry – reinforcement definition in absolute local wall coordinates.....	32
Figure 29: Dialog for reinforced masonry – definition of horizontal reinforcement.....	33

Figure 30: Dialog for reinforced masonry – advanced properties	34
Figure 31: Dialog for reinforced concrete material – standard sheet	35
Figure 32: Dialog for reinforced concrete material – vertical reinforcement definition.....	36
Figure 33: Dialog for reinforced concrete material – reinforcement definition in absolut local wall coordinates	37
Figure 34: Dialog for reinforced concrete material – definition of horizontal reinforcement	38
Figure 35: Spandrel/Lintel input dialog.....	39
Figure 36: RC ring input dialog	40
Figure 37: Local coordinate system of the wall and the wall finite element.....	42
Figure 38: Description of various wall materials in AmQuake.....	42
Figure 39: Example of a layer for walls.....	44
Figure 40: Example of a layer for windows.....	44
Figure 41: Example of a layer for a ceiling	45
Figure 42: Example of a wall or window definition in a DXF file	45
Figure 43: Example of wall, windows and ceiling definition in a DXF file.	46
Figure 44: Possible definition of connection between windows and walls	46
Figure 45: Import DXF - walls.....	47
Figure 46: Import DXF - windows	49
Figure 47: Import DXF - ceiling	51
Figure 48: Topology – Walls / Columns	53
Figure 49: Topology – Windows / Doors.....	54
Figure 50: Topology – Ceilings.....	55
Figure 51: Ceiling Geometry and Loading Areas	55
Figure 52: Ceiling load.....	58
Figure 53: Generate Mesh.....	60
Figure 54: Generate Mesh Dialog when automatic Conversion of CM to RM model is selected	60
Figure 55: Mesh - Nodes.....	61
Figure 56: Mesh - Elements	62
Figure 57: Mesh – Rigid lines	63
Figure 58: Calculate – Analyses.....	64
Figure 59: Calculate – Advanced Settings	65
Figure 60: Results – Analyses	68
Figure 61: Results – Analyses – Table with results.....	69


Figure 62: Summary of the Pushover Analysis Evaluation	71
Figure 63: Dialog for the Setting of various Visualization Options. This dialog is opened using the button 	72
Figure 64: Results – Analyses – Analysis 2	73
Figure 65: Selection of Deformed Shape and Visualization of Damaged Elements	73
Figure 66: Summary of Analysis Results	74
Figure 67: Results: LD diagram.....	75
Figure 68: Results: Capacity / Target diagram.....	75
Figure 69: Results: Se vs. norm displacements	76
Figure 70: Results: Forces	76
Figure 71: Results: Reactions.....	77
Figure 72: Results: Displacements	77
Figure 73: Results: Components damages	78
Figure 74: Results: M-N diagram.....	78
Figure 75: Print statement	79
Figure 76: Model RM - modelling of confined masonry using a single wall with edge reinforcement.....	81
Figure 77: Model CM - modelling of confined masonry by two columns and one middle wall. The columns can be assigned either a reinforced masonry material or reinforced concrete material.	81
Figure 78: Automatic conversion from CM to RM model, (a) original model, (b) CM model, (c) automatically converted RM model. The automatic conversion is activated by the parameter <code>Change PM and RC to RM</code> in the <code>Advanced Settings</code> of the <code>Mesh</code> dialog.	82
Figure 79: A model of a confined masonry building, which using the confined masonry model according to the method (c, i.e. Model CM).....	83
Figure 80: DXF file with the definition of confined regions for the confined masonry building.....	83
Figure 81: DXF file definition with the separate layers for masonry elements and confining elements.	84
Figure 82: Definition of vertical reinforcement for reinforced masonry	85
Figure 83: Definition of horizontal reinforcement for reinforced concrete masonry	85
Figure 84: Analysis with convergence problems and high errors.....	86
Figure 85: Reinforced concrete ring definition.....	87
Figure 86: Continuous r.c. ring is generated for neighbouring walls and windows/doors with identical orientation of the local z axis.	88

Figure 87: R.C. ring is generated through the corner of the building if the corner walls are defined such that they slightly overlap each other, and the center of their side faces is close to each other.	89
Figure 88: Shape of the elastic response spectrum	91
Figure 89: Definition of an idealized relationship.....	95
Figure 90: Structural model of a masonry wall disregarding (top) and considering (bottom) masonry lintels or parapets (Reference [1]).....	99
Figure 91: A 3D model of structure	100
Figure 92: Wire model of a structure	101
Figure 93: Rigid link.....	101
Figure 94: Vertical connection of walls.....	103
Figure 95: Idealized diagram for masonry wall shear behavior.....	110
Figure 96: Idealized diagram for reinforced concrete shear behavior.....	113
Figure 97: AmQuake beam subject to longitudinal and lateral loading.....	114
Figure 98: Idealized bi-linear behaviour of the cross-section in bending.	117
Figure 99 M-N diagram of a cross section.	118

LIST OF TABLES

Table 1: Importance classes for buildings for Eurocode 8.....	17
Table 2: Importance classes for buildings for Romania P100.....	17
Table 3: Masonry unit data	105
Table 4: Mortar Properties	106

1 INTRODUCTION

AmQuake is a software program for seismic assessment of masonry buildings. The seismic assessment is performed according to Eurocode 8 and 6 using the pushover analysis and equivalent frame method.

In addition the program supports also a static analysis of masonry building according to the Eurocode 6.

AmQuake program includes national specifics as they are defined in corresponding annexes to EC6 and EC8 for the supported countries. In addition the program supports Romanian standards P100 and CR6. The national specific parameters for the various formulas implemented in the program are defined by the national coefficient table, which is printed in Appendix A of this document.

The program development was supported by Wienerberger A.G and therefore it includes complete database of Wienerberger masonry products.

2 INSTALLATION AND REGISTRATION

The program is installed from a single installation file that can be distributed on a CD, usb memory stick or can be direction downloaded from the internet. The latter approach is preferable as it ensures that always the latest program version is installed. The internet address with the latest information about the program is:

www.amquake.eu

In the section downloads at this internet address, the latest available program version can be found and downloaded. The program includes support for the following national localizations:

Austria
Belgium (Dutch and French)
Bulgaria
Croatia
Czech Republic
France
Greece
Hungary
Romania EN 1996, 1998
Romania CR6, P100 (2006)
Romania CR6, P100 (2013)
Slovakia
Slovenia
Italy

The program is not protected by any hardware lock, but after the program installation a registration is required.

2.1 Installation process

Before starting the installation process you should verify if you have administrator privileges on your computer. It will not be possible to install the program with normal user privileges.

Step 1: Download the installation file to your computer or insert the program CD into your CD drive. Click on the installation file using the right mouse button and select the option: **Run as Administrator**.

Step 2: After that the installation file is verified and the installation process will start with the following dialog for the selection of your national language.

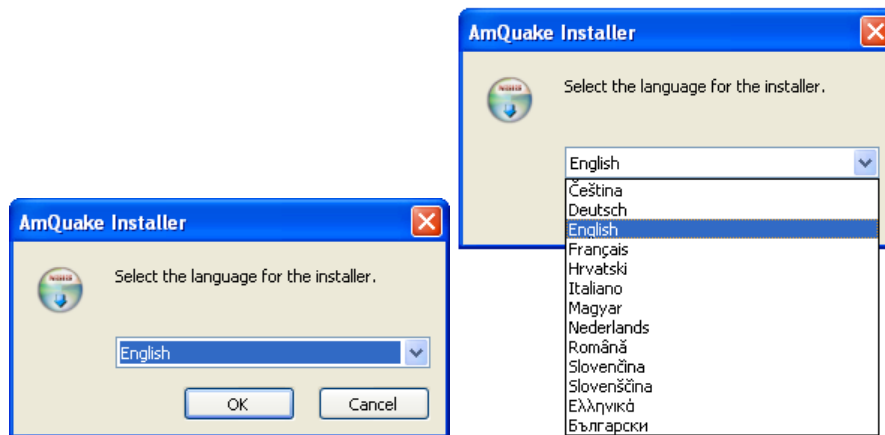


Figure 1: Selection of national language during the installation process

It is recommended to select your country language at this point as this will be the default language for the program after the installation. During the program execution it is possible to change the national language always when a new file is created.

The next dialog (Figure 2) allows you to define the program components that will be installed. If there is no pdf reader available on your computer, the item **PDF Acrobat Reader** should be selected. This reader is necessary for viewing the program documentation as well as the Wienerberger product sheets.

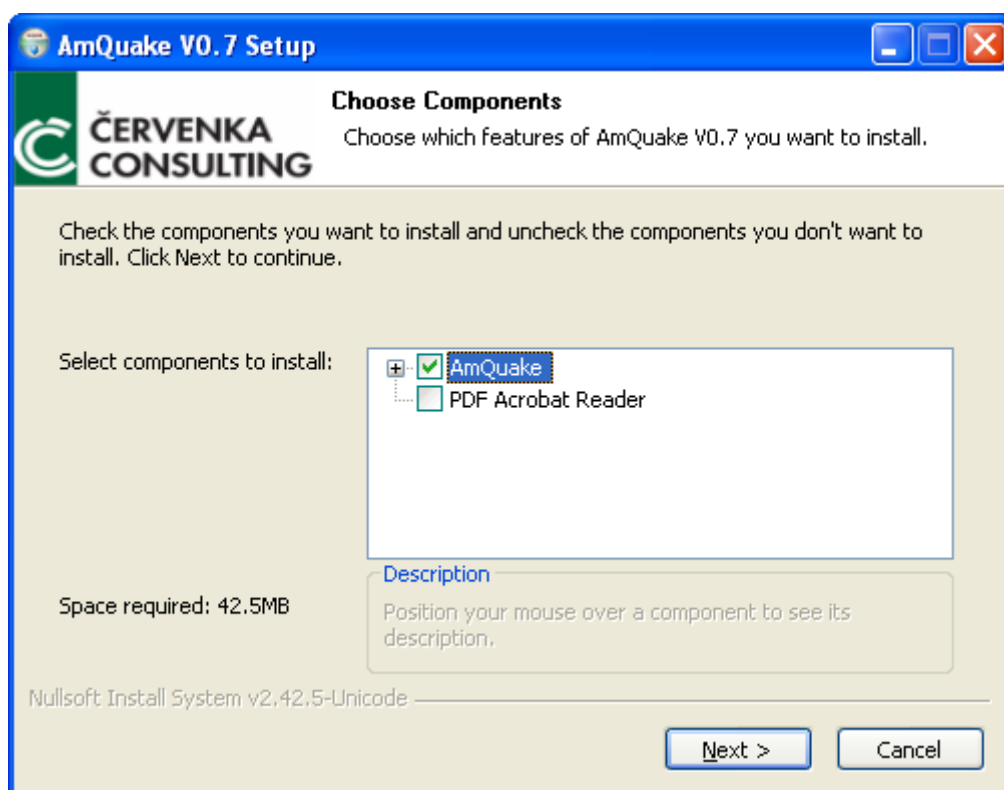


Figure 2: Selection of program components to be installed

After clicking the next button the program will be installed on your computer. Typically the program is installed in the **Program Files** directory on your root hard drive. In this directory a subdirectory **Cervenka Consulting/AmQuake** will be created. This directory contains the program executable, documentation as well as the example files. **It is recommended to copy the example files into a different directory before trying to open them with AmQuake.**

After the installation is completed the program **AmQuake** can be executed. You can find the program in your **Start menu** under the item **CervenkaConsulting**.

Right after the installation, the program will be operating only in a demo regime. In order to use the full version of the program it is necessary to register it, and obtain a permanent program password. This is explained in the next paragraph of this document.

2.2 Registration process

In order to perform a successful program registration it is necessary to satisfy the following conditions:

What you need:

- (1) **internet connection** from the computer where the program will be installed
- (2) have a **valid licence number**. This number can be purchased online from the above internet address or by contacting the program distributor. Limited number of free licences is available at your local Wienerberger offices. If you are interested in the free program licence, you should contact your local Wienerberger office for more information.
- (3) in addition you will need also a **usb memory stick**. It should be a stick that is compatible with the program **AmQuake**. This list of compatible usb memory sticks are listed at the end of this section. This memory stick needs to be attached to your computer during the registration process. After the successful registration you can use this memory stick as your own personal hardware key for the program **AmQuake**. This means it needs to be attached to your computer every time **AmQuake** is executed. Without this the program **AmQuake** is executed only in demo mode, which is limited to 10 walls. This usb key can be however used on other computers as well. This means that you can use **AmQuake** on any computer while your registered usb stick is attached to it.

*For **student version** of **AmQuake** you don't need an usb memory stick. But in the dialog **License activation** (Figure 4) usb device task, you must select **Student version**. The registration process for the Student version is described in detail in Section 2.3).*

The registration process is started by selecting the menu item **Help | About ATENA Masonry Quake**. This opens the following dialog, in which the button **License activation** should be selected. Please make sure that your usb memory stick that you want to use for **AmQuake** registration is attached to your computer before selecting this button.

The activation process is described by the subsequent figures.

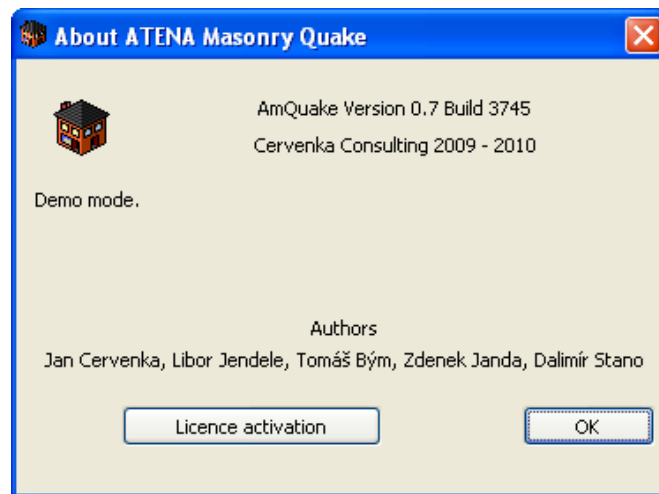


Figure 3: The About dialog with the licence activation button.

License activation

Usb device: {Select USB drive}
 USB#: (Select USB drive)
 Company:
 License Version: 0.7.3747 modif.
 Serial number: - - - -
 Registration key:
 Go to web registration
 Write license to usb Cancel

In this pull down box it should be possible to select your usb drive.

!If it does not appear here, most probably it is an incompatible usb drive and you should try to use another one.

License activation

Usb device: E: (kingston)
 USB#: 5B8306001248
 Company: Cervenka Consulting
 License Version: 1.0.3812 modif.
 Serial number: CLW74 - MMW3G - KIA9C - IDOLZ - X685N
 Registration key:
 Go to web registration
 Write license to usb

A valid license should be filled in these fields and then use the **Go to web registration** button to obtain the registration key.

Figure 4: License activation dialog

ČERVENKA CONSULTING

AMQUAKE License Activation

Step 1/3

Serial key:

Name:

Version:

USB#:

Email:

☒ Yes, I want to receive AmQuake newsletter.

Insert your email address, then select if you want to receive AmQuake newsletter in the future.

Finalize this step by pressing the „Validate license“ button.

Figure 5: AmQuake online license activation, step 1/3

ČERVENKA CONSULTING

AmQuake License Activation

Step 2/3

Your license is valid.

Name: **Cervenka Consulting**

Is your name displayed correctly?

This is the confirmation of your company name. Please, note that this name will appear on all reports and figures created by AmQuake program.

Continue with the activation process by pressing this button.

Figure 6: AmQuake online license activation, step 2/3

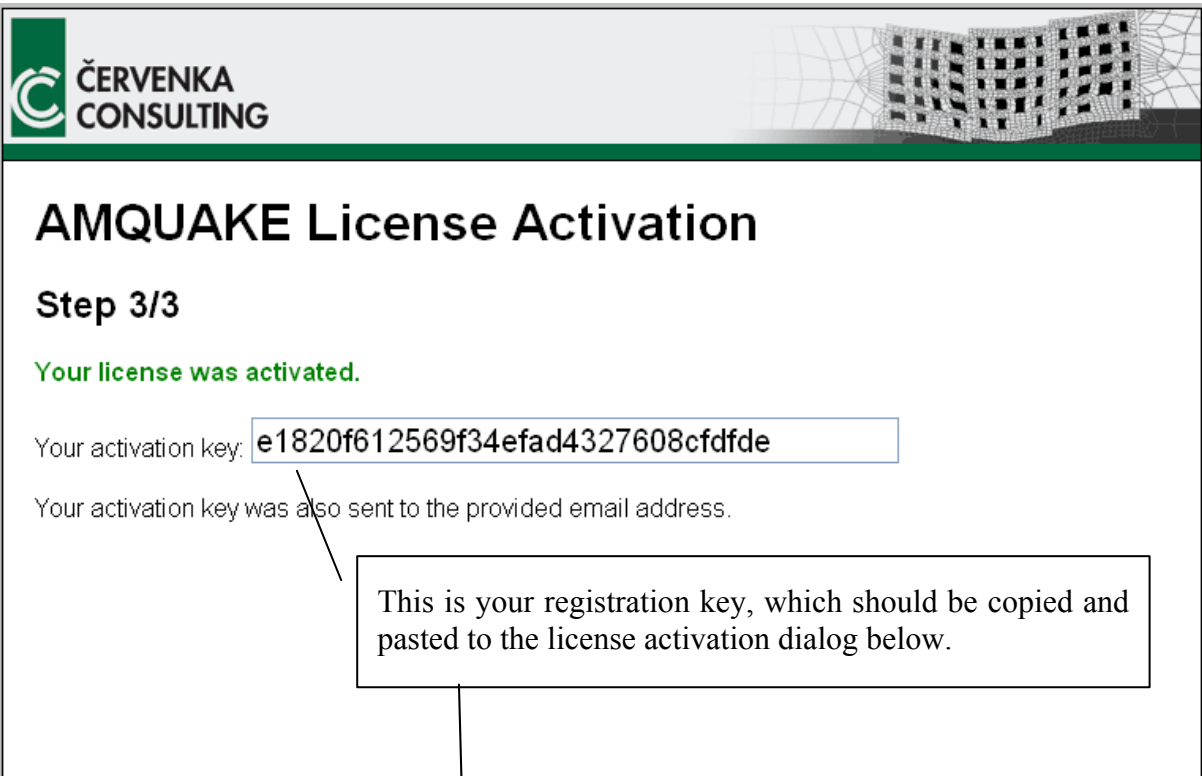


Figure 7: AmQuake online license activation, step 3/3

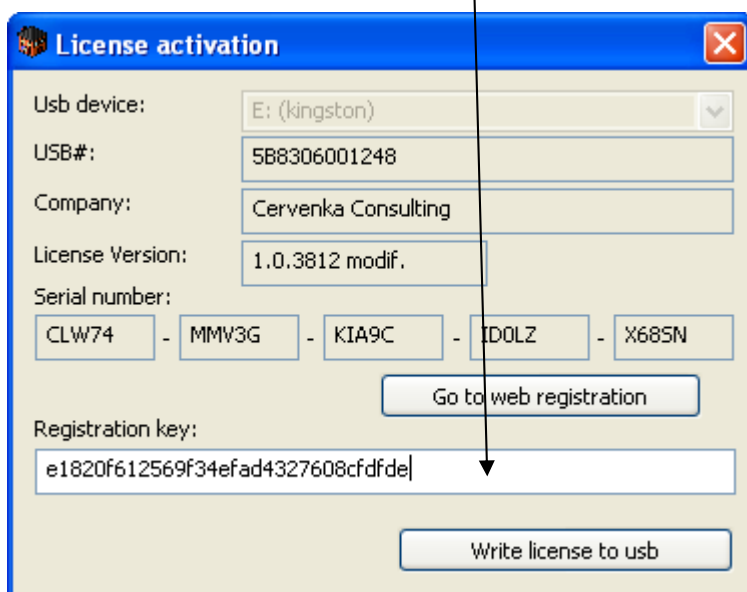


Figure 8: Final step in the program activation

The final step in the activation process is to copy the registration key from the **AmQuake** registration website. Alternatively, the registration key will be sent to you also by a separate email.

In case of problems, please contact **AmQuake** support staff by sending an email to

support@amquake.eu

2.3 Registration process for student (university) version

In case of student version, the registration is very similar, but you do not need have a usb memory stick for registration. During the license activation you need to select Student version. Specify the Serial number and click to Go to web registration. After the activation on the web, you have to input registration key and company name. You will receive get both information from our website, after the registration process is completed. Also you will receive an e-mail.

Student version is like a full version, but with the following restrictions:

- In all images and graphs the name of the University is printed.
- In the exported analysis protocol, only first 10 rows of results is available for each result table.

All data files created by the student version will have the above restrictions, even if they are opened and recalculated in the full version of the program.

Figure 9: Final step in the program activation – student version

3 USER'S MANUAL

This chapter contains the description of the graphical user interface of the program **AmQuake**. Each window of the program as well as each input dialog will be described here along with the necessary input parameters.

AmQuake software uses the following basic set of units:

m, kN, ton, kPa

with the exception of stress and strength units, these will be specified on the user input in MPa or N/mm^2 depending on the national localization. However, these units are always converted to the above basic set for the numerical analysis.

Another exception are weight units, these will be specified by the user in kg and then converted to the above basic set.

The main program window is described in Section 3.1. This description of the user interface is following the order of the menus and items that are visible on this main program screen starting from top left corner.

3.1 Main window

The application main window is divided into 4 panels including the menu strip and space for toolbars on the top. The user is guided through the analysis by the tree in the panel on the left side. Actual information appears in the bottom panel. The main and also the largest panel contain the rendering windows for render the geometry.

User starts with creating new project and follows the step-by-step options in the tree.

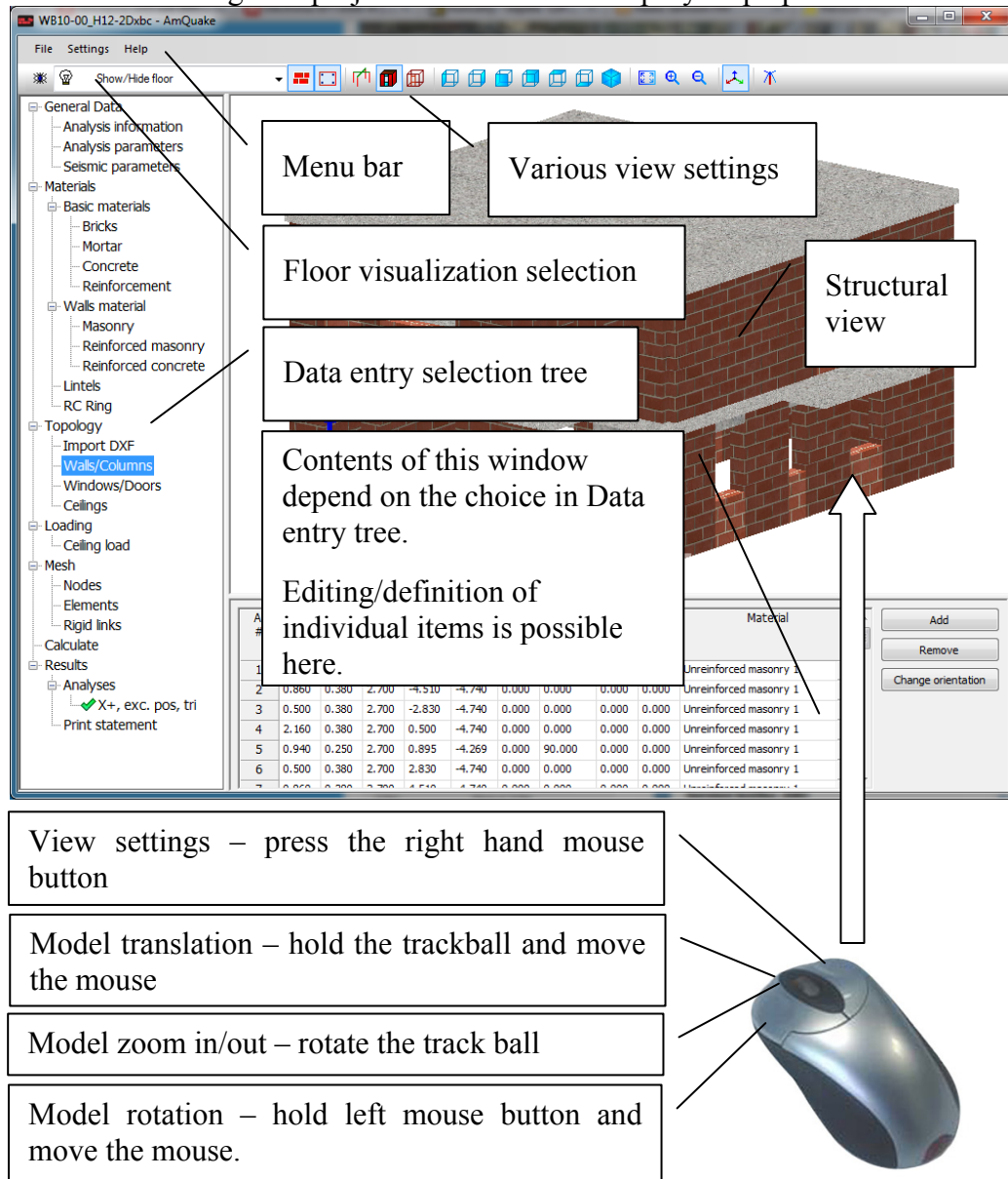


Figure 10: AmQuake screen and description of model zoom and rotation using the mouse buttons

3.2 Main menu bar

The main menu bar contains the following items:

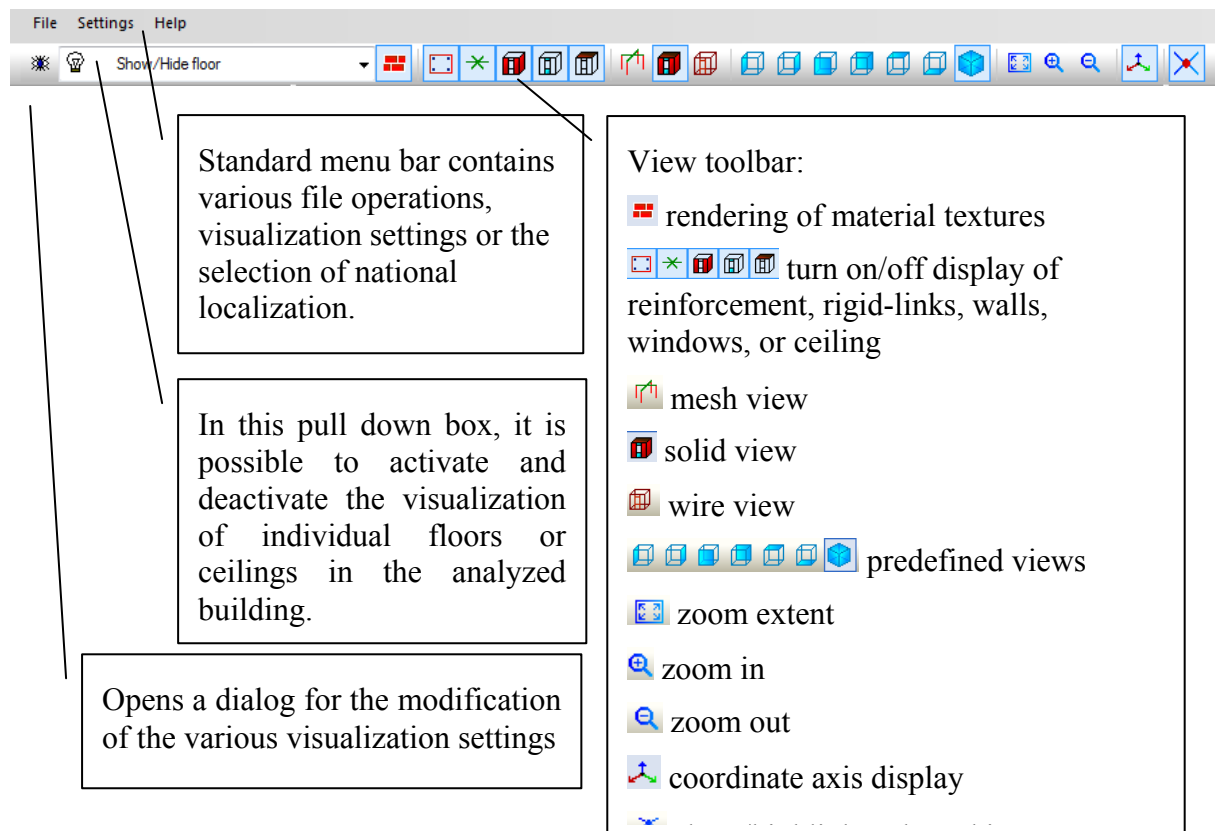


Figure 11: Main menu bar

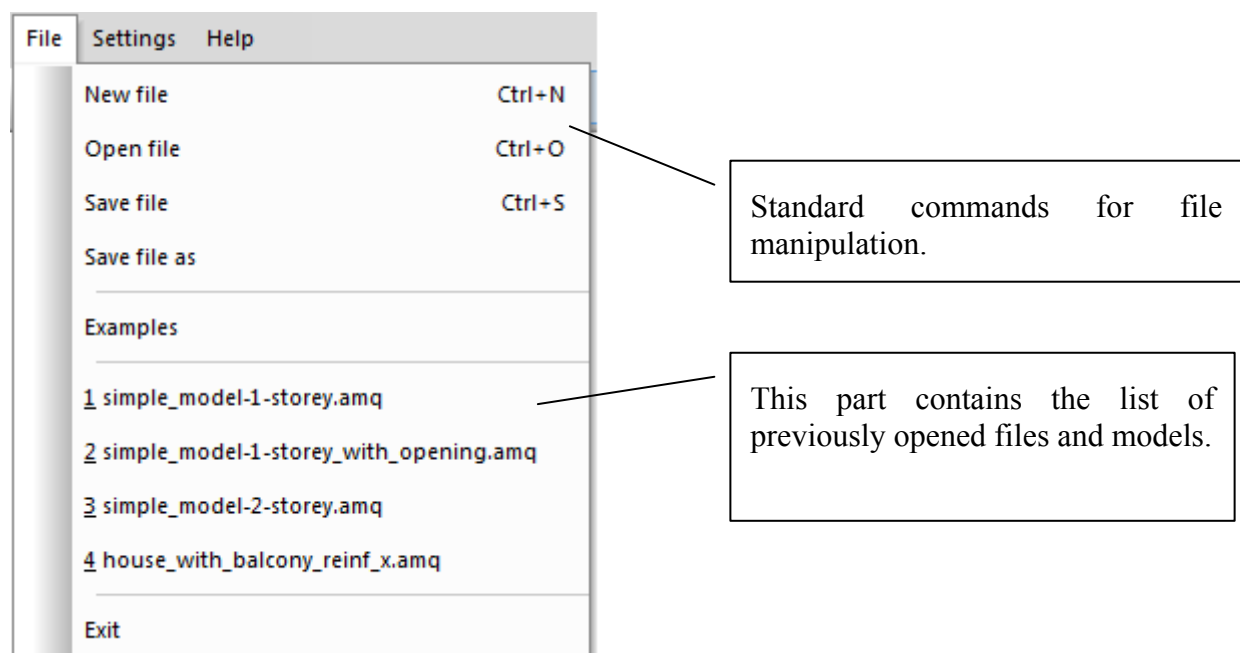


Figure 12: File menu items

The standard for each country can be selected when a new file is created as it is shown in Figure 13.

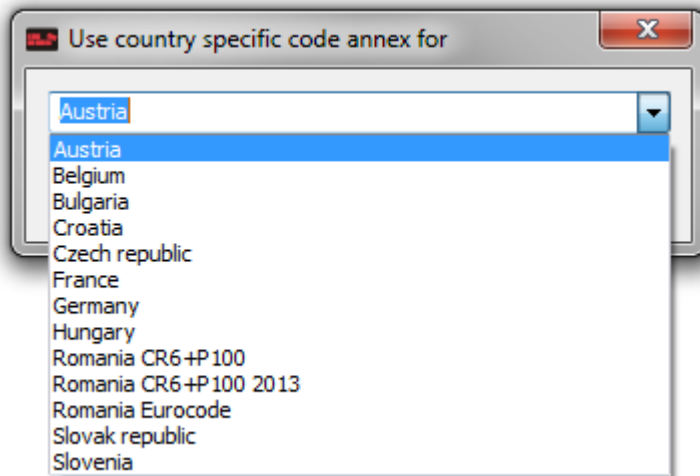


Figure 13: Selection of the country specific standard from the menu File | New file.

The menu Settings is used to select the national language for the program as well as the standard or advanced user mode. The choice of national localization does not affect the national standard, but it only affects the language that the program is using for the communication with the user.

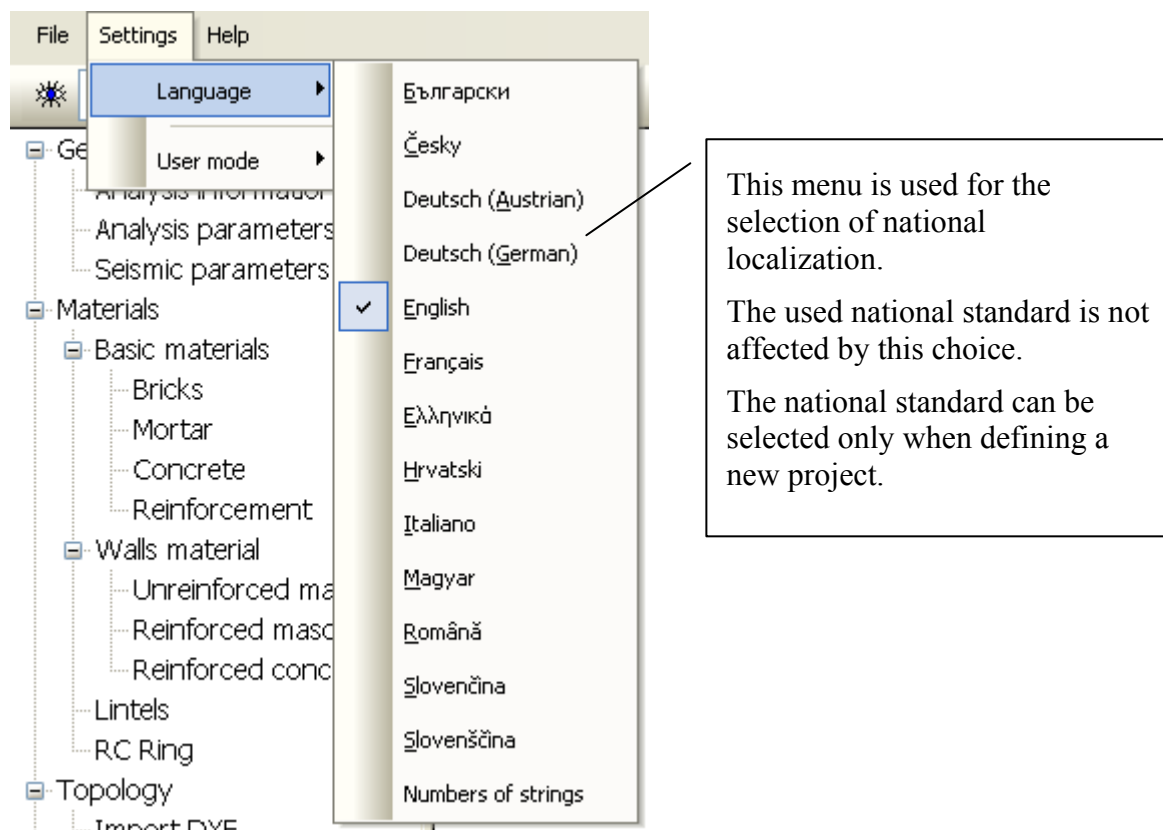


Figure 14: Settings menu items

The user mode switches between the standard mode, which provides an access to all features of the program. An alternative is the simple user mode, which can be used for the basic program operation. This is recommended for the users that do not want to exploit the

extensive possibilities of various program settings. This version also provides only a limited set of most important relevant results for post-processing and analysis evaluation.

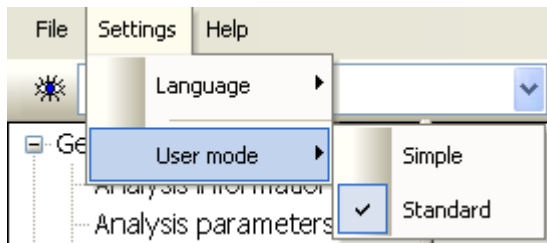


Figure 15: Selection of simple and standard user mode

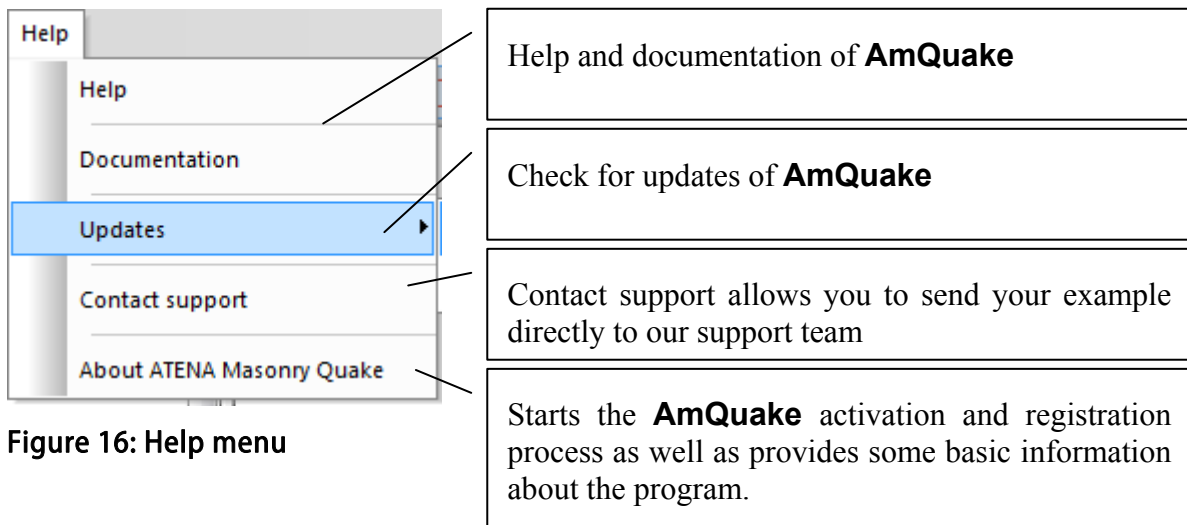


Figure 16: Help menu

3.3 General data

General data is the first item in the main data entry selection tree. It is used to define the project description as well as main analysis and seismic parameters

3.3.1 Analysis information

This item allows the user to specify general information about the project such as name of the company, description of the project and date. User opens the input dialog window by clicking **Edit** button.

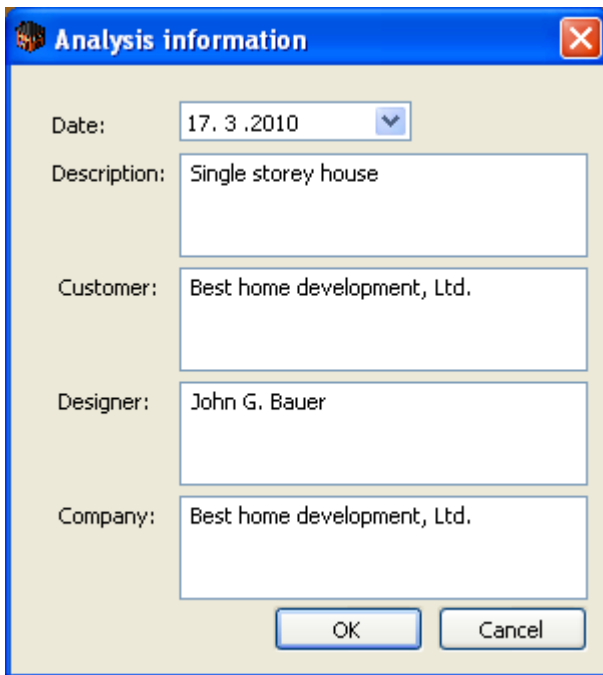


Figure 17: Analysis information

Date	= date, will be shown in the analysis protocol
Project description	= project description, it will appear in the analysis protocol
Customer	= name of the client, for which the analysis is being done.
Designer	= architect and designer of the building to be analyzed.
Company	= name of the company responsible for this analysis

3.3.2 Analysis parameters

This input data tree item activates the definition of various global analysis parameters. The parameters are defined within the dialog, which appears at the bottom part of the program screen. There are two tab controls in the dialog. On the first tab – **Building standard**, the currently active building standard is shown according to the selected national localization. On the second tab **Advanced settings** it is possible to manually set various parameters.

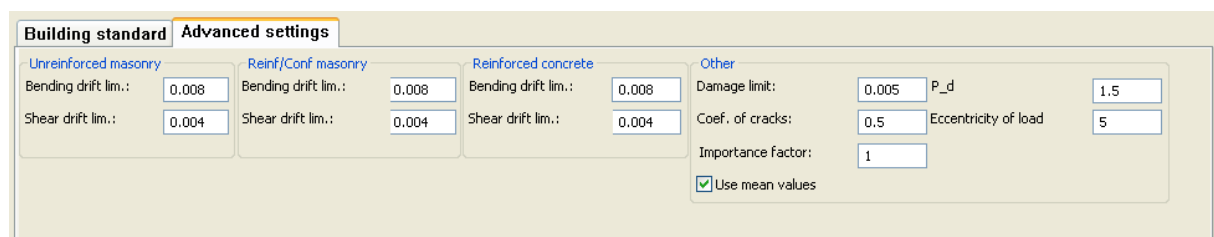
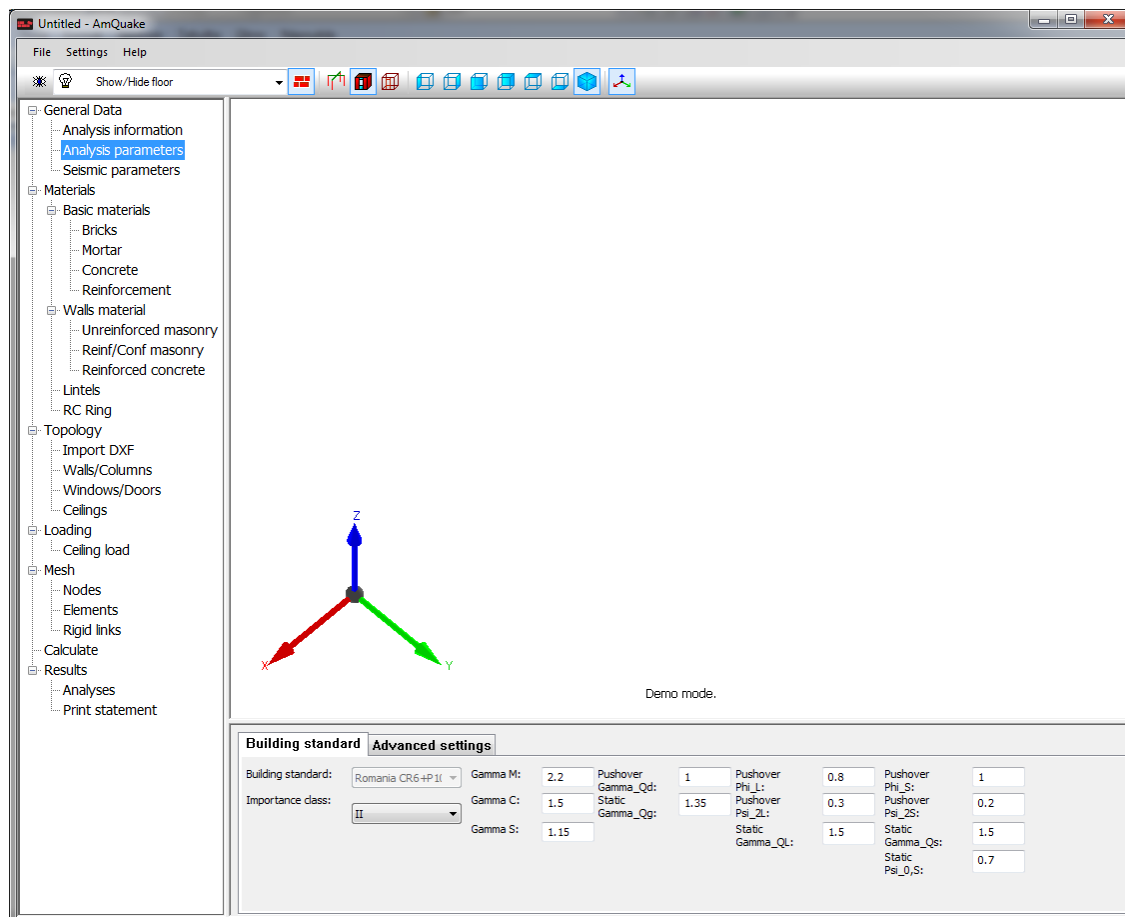


Figure 18: Analysis parameters – Building assessment standard

Building standard tab:

Building standard = here the currently active building standard is shown. The program supports the following national localizations of Eurocode 2, 6 and 8:

Austria
Belgium
Bulgaria
Croatia
Czech Republic
France
Germany

Greece

Hungary

Romania CR6+P100 2006

Romania CR6+P100 2013

Romania Eurocode

Slovakia

Slovenia

(Some of the national standards are not yet supported, but will be supported in the future. Please check your current program version for the list of supported standards)

Building importance class = determines the importance factor for the analyzed building. The importance classes are defined in EN 1998-1, Section 4.2.5, Table 4.3.

Table 1: Importance classes for buildings for Eurocode 8

Importance class	Buildings
I	Buildings of minor importance for public safety, e.g. agricultural buildings, etc.
II	Ordinary buildings, not belonging in the other categories.
III	Buildings whose seismic resistance is of importance in view of the consequences associated with a collapse, e.g. schools, assembly halls, cultural institutions etc.
IV	Buildings whose integrity during earthquakes is of vital importance for civil protection, e.g. hospitals, fire stations, power plants, etc.

Table 2: Importance classes for buildings for Romania P100

Importance class	Buildings
I	Buildings with essential functions, the integrity of which during earthquakes is vital for civil protection: fire and police stations, hospitals, and other constructions afferent to sanitary services being endowed with emergency and surgery rooms; buildings of institutions with responsibility in handling the emergency cases related to national defense and security; stations of producing and distributing energy and/or provides essential services for the other types of buildings mentioned above; garages of emergency services or different categories; water supply and water pumping stations for emergencies cases; buildings sheltering toxic gases, explosives and other dangerous substances.

II	<p>Buildings with important seismic resistance due to consequences associated to collapse or serious damage:</p> <ul style="list-style-type: none"> • Dwelling and public buildings with more than 400 persons in the total exposed area • Hospitals other than I class hospitals and medical institutions with the capacity exceeding 150 persons in the total exposed area. • Prisons • Asylums and day nursery • Different grade schools with a capacity of exceeding 200 persons in total exposed area • Auditoriums, conferences and showrooms with a capacity exceeding 200 persons <p>Museums, national importance buildings</p>
III	Current type buildings that are not included in the other categories.
IV	Less important buildings for public safety with low occupancy degree and/or less economic importance ,agricultural constructions

Gamma_M: Partial safety factor γ_M for masonry.

Gamma_C: Partial safety γ_C factor for concrete.

Gamma_S: Partial safety factor γ_S for steel.

It should be noted that partial safety factors for the materials that are specified in Figure 18 are used only in the static analysis. The pushover analysis is performed using average material properties, i.e. the above factors are set automatically by the program to 1.0 for the pushover analysis.

Pushover Gamma_Gp: Multiplier $\gamma_{G,p}$ for permanent load G for pushover analysis. In most national standards this multiplier should be equal to 1.0.

Static Gamma_Gs: Multiplier $\gamma_{G,s}$ for permanent load G for static analysis.

Pushover Phi_L: Multiplier φ_L for live load Q_L for pushover analysis.

Pushover Psi_2L: Multiplier ψ_{2L} for live load Q_L for pushover analysis.

Static Gamma_QL: Multiplier γ_{QL} for live load Q_L for static analysis.

Pushover Phi_S: Multiplier φ_s for snow load Q_s for pushover analysis.

Pushover Psi_2S: Multiplier $\psi_{2,s}$ for snow load Q_s for pushover analysis.

Static Gamma_Qs: Multiplier γ_{Qs} for snow load Q_s for static analysis.

Static Psi_0,S: Multiplier $\psi_{0,s}$ for snow load Q_s for static analysis. The following load combinations are used:

The load combination for the pushover analysis is:

$$\gamma_{Gd} G + \psi_{EL} Q_L + \psi_{Es} Q_s \quad (1.1)$$

where:

$$\psi_{EL} = \varphi_L \psi_{2,L}, \quad \psi_{Es} = \varphi_S \psi_{2,s}$$

and the load combination for the static analysis is:

$$\gamma_G G + \gamma_{QL} Q_L + \gamma_{QS} \psi_{0,s} Q_s \quad (1.2)$$

Advanced Tab:

In this dialog, it is possible to modify various parameters related to the pushover analysis. It should be noticed that some of the input parameters that are defined in the advanced tab may depend on the values from the base tab. This means that they may change depending on the changes of the base tab parameters. For instance the value of the importance factor depends on the importance class of the building, so the importance factor changes when different importance class is selected.

Ordinary masonry

Bending drift limit NCR d_{lim}^M = drift limit for bending failure. The value defines the relative drift for each wall when the load carrying capacity in bending is fully exhausted, i.e. the ultimate moment drops to zero. This value depends on national annex and typically is equal to 0.008. This means that ultimate moment drops to zero when

$$\frac{\Delta u}{H} \geq d_{lim}^M \quad (1.3)$$

where Δu is the relative displacement of the element end points, H height of the wall, i.e. the length of the investigated element.

Shear drift limit NCR d_{lim}^V = drift limit for shear failure. The value defines the relative drift for each wall when the load carrying capacity in bending is fully exhausted, i.e. the ultimate moment drops to zero. This value depends on national annex and typically is equal to 0.004. This means that ultimate shear force drops to zero when

$$\frac{\Delta u}{H} \geq d_{lim}^V \quad (1.4)$$

where Δu is the relative displacement of the element end points, H height of the wall, i.e. the length of the investigated element.

Reinforced masonry

Bending drift limit NCR = drift limit for bending failure. The value defines the relative drift for each wall when the load carrying capacity in bending is fully

exhausted, i.e. the ultimate moment drops to zero. This value depends on national annex and typically is equal to 0.008. (See Equation (1.3))

Shear drift limit NCR = drift limit for shear failure. The value defines the relative drift for each wall when the load carrying capacity in bending is fully exhausted, i.e. the ultimate moment drops to zero. This value depends on national annex and typically is equal to 0.004. (See Equation (1.4)).

Confined Masonry

Bending drift limit NCR = drift limit for bending failure. The value defines the relative drift for each wall when the load carrying capacity in bending is fully exhausted, i.e. the ultimate moment drops to zero. This value depends on national annex and typically is equal to 0.008. (See Equation (1.3))

Shear drift limit NCR = drift limit for shear failure. The value defines the relative drift for each wall when the load carrying capacity in bending is fully exhausted, i.e. the ultimate moment drops to zero. This value depends on national annex and typically is equal to 0.004. (See Equation (1.4)).

Reinforced Concrete

Bending drift limit NCR = drift limit for bending failure. The value defines the relative drift for each wall when the load carrying capacity in bending is fully exhausted, i.e. the ultimate moment drops to zero. This value depends on national annex and typically is equal to 0.008. (See Equation (1.3))

Shear drift limit NCR = drift limit for shear failure. The value defines the relative drift for each wall when the load carrying capacity in bending is fully exhausted, i.e. the ultimate moment drops to zero. This value depends on national annex and typically is equal to 0.004. (See Equation (1.4)).

Other

Damage limit DLR d_{lim} = inter-storey drift limit for DLS evaluation. This value depends on the national annex, and typically should be equal to 0.005. The program checks if the maximal inter-storey drift is below this limit when the master displacement (i.e. top displacement) reaches the DLS target value.

$$\frac{\Delta u}{h} \leq d_{lim} \quad (1.5)$$

where d_r is the inter-storey drift, i.e. relative horizontal displacement of one storey, and h is the storey height.

Coef. of cracked wall stiffness = This parameter defines the reduction of initial wall stiffness due to the tensile cracking. The recommended value is 0.50 (see EN 1998-1, Section 4.3.1 (7)).

Importance factor γ_I = this value depends on Building importance class and it is usually defined in the national annex. If not defined otherwise the recommended values for Eurocode 8 are I – 0.8, II – 1.0, III – 1.2, IV – 1.4

Use mean values = when this checkbox is selected mean values of material properties are used for the pushover analysis. In most cases, this checkbox should be selected as EN 1998-1 (Section 4.3.3.4.1 (4)) requires that mean values are used in pushover analysis.

P_d = This value represents a safety factor for pushover analysis. The calculated displacement capacity should be larger than the target displacement multiplied by this value.

Eccentricity of load = This value represents the percentual eccentricity e_a of the horizontal forces in the pushover analysis. In most national standards, it is equal to 5%.

$$e_{ai} = \pm \frac{e_a}{100} L_i \quad (1.6)$$

where L_i is the dimension of the i-th floor perpendicular to the direction of the seismic action.

3.3.3 Seismic parameters

This input data tree item activates the definition of seismic parameters. The seismic parameters depends on the location of the analyzed building and on the foundation properties.

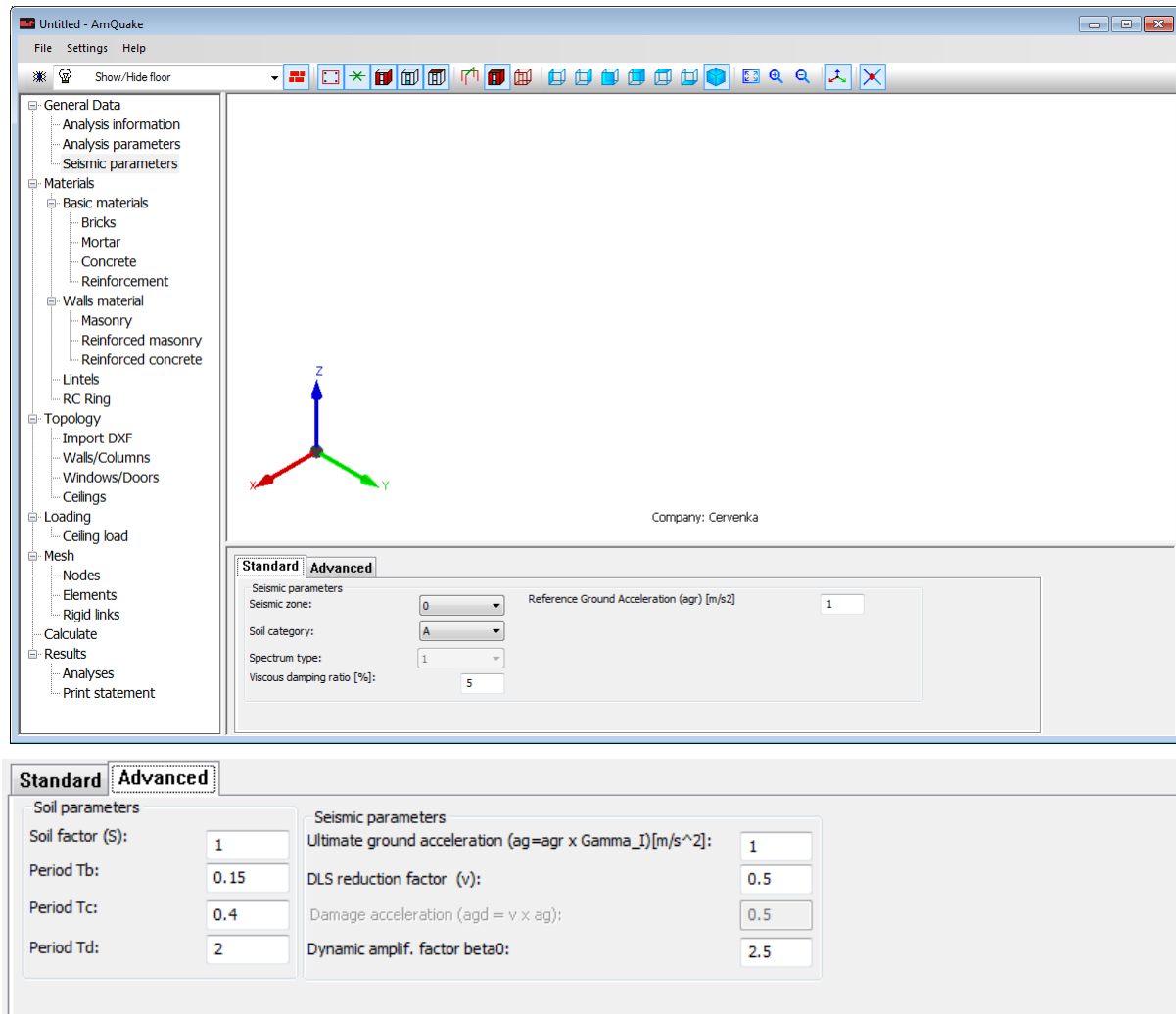


Figure 19: Seismic parameters

Standard tab:

Seismic zone : in certain European countries, each seismic region is categorized according to the expected reference ground acceleration into several seismic zones. If this is the case, an appropriate zone is to be selected here. This seismic zone selection influences the final value of the importance factor (see Section 3.3.2)

Soil category : ground types are described by stratigraphic profiles based on the shear wave velocity $v_{s,30}$, standard penetration test blow count N_{SPT} and undrained shear strength of the soil c_u .

Ground type	Description of stratigraphic profile	Parameters		
		$v_{s,30}$ (m/s)	N_{SPT} (blows/30cm)	c_u (kPa)
A	Rock or other rock-like geological formation, including at most 5 m of weaker material at the surface.	> 800	–	–
B	Deposits of very dense sand, gravel, or very stiff clay, at least several tens of metres in thickness, characterised by a gradual increase of mechanical properties with depth.	360 – 800	> 50	> 250
C	Deep deposits of dense or medium-dense sand, gravel or stiff clay with thickness from several tens to many hundreds of metres.	180 – 360	15 - 50	70 - 250
D	Deposits of loose-to-medium cohesionless soil (with or without some soft cohesive layers), or of predominantly soft-to-firm cohesive soil.	< 180	< 15	< 70
E	A soil profile consisting of a surface alluvium layer with v_s values of type C or D and thickness varying between about 5 m and 20 m, underlain by stiffer material with $v_s > 800$ m/s.			
S_1	Deposits consisting, or containing a layer at least 10 m thick, of soft clays/silts with a high plasticity index ($PI > 40$) and high water content	< 100 (indicative)	–	10 - 20
S_2	Deposits of liquefiable soils, of sensitive clays, or any other soil profile not included in types A – E or S_1			

Figure 20: Ground types classification table (NOTE: this table may be specified for each country in its national annex)

For Romanian code P100 this parameter is not used. Instead the country is divided into several zones depending on value of the period T_C . Based on that in Romania three zones are recognized: Z1: $T_C = 0,7$ s, Z2: $T_C = 1,0$ s, Z3: $T_C = 1,6$ s,

Spectrum type : generally Eurocode 8 recognizes two type of elastic spectra (1 and 2I) that can be used in a seismic assessment. The recommended choice of the elastic spectra is defined in the national annex for each country.

Viscous damping ratio : this parameter defines the viscous damping ratio of the structure given in %. The recommended value for Eurocode is typically 5.0 %, but generally it depends on the national standard. It is used to calculate the damping correction factor η , which is a necessary parameter for the elastic response spectrum formulas.

$$\eta = \sqrt{10 / (5 + \xi)} \geq 0.55 \quad (1.7)$$

Reference ground acceleration : a_{gR} , this parameter defines the reference ground acceleration in m/s^2 on type A ground as specified by the national annex for each country. The reference ground acceleration is usually specified for each country for each region or city. *Please note that total value of a_{gR} in m/s^2 must be defined here. If your country annex defines the ground acceleration as a relative value, it should be first multiplied by $g = 9,81 \text{ m/s}^2$.*

Advanced tab

The parameters in this input window should be modified with extreme caution. They are automatically generated by the program based on the various parameters defined on the basic property sheet.

Soil factor : soil factor for the elastic response spectrum, this value is automatically generated by the program based on the ground type and elastic spectrum type. It is not recommended to modify this value. For certain national standards such as Romania EC8 and P100 this value should be set to 1.0.

Period T_b : this is the lower limit of the period with constant acceleration in the elastic response spectrum. This value is automatically generated by the program based on the ground type and elastic spectrum type. It is not recommended to modify this value.

Period T_c : this is the upper limit of the period with constant acceleration in the elastic response spectrum. This value is automatically generated by the program based on the ground type and elastic spectrum type. It is not recommended to modify this value.

Period T_d : this is the lower limit of the period with constant displacement in the elastic response spectrum. This value is automatically generated by the program based on the ground type and elastic spectrum type. It is not recommended to modify this value.

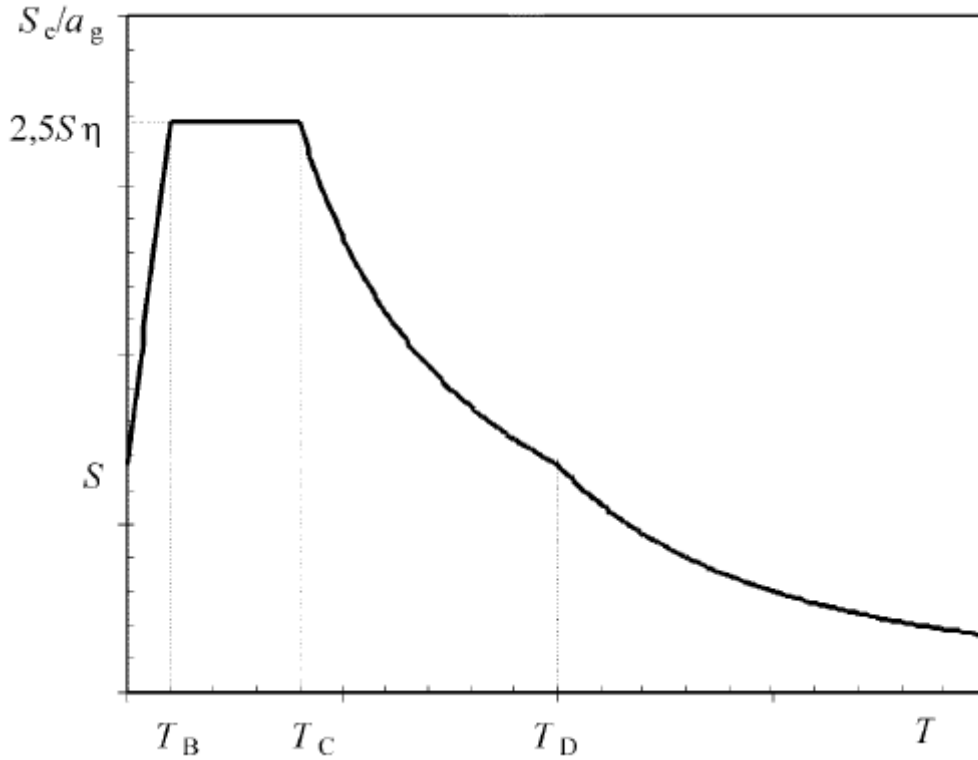


Figure 21: Typical shape of the elastic response spectra, (see Section 5.1).

Ultimate ground acceleration : this value is in Eurocode 8 denoted as a_g , and it corresponds to the design ground acceleration for Ultimate Limit State (ULS) verification used in the elastic response spectra formulas. This value is automatically calculated by the program as:

$$a_g = \gamma_I a_{gR} \quad (1.8)$$

where:

γ_I -importance factor, topographic amplification factor for important structures (see **Analysis parameters | Advanced**)

a_{gR} - reference peak ground acceleration on type A ground (see **Seismic parameters | Standard**)

Damage acceleration – a_{Dg} , this acceleration value is used to calculate target displacement for damage limitation state (DLS). The basis for the calculation of a_{Dg} is the “damage limitation requirement” of probability of exceedance $P_{DLR} = 10\%$ in 10 years.

$$a_{Dg} = \gamma_D \gamma_I a_{gR}, \quad \text{where } \gamma_D = \left(\frac{T_{LR}}{T_L} \right)^{-\frac{1}{k}} = \left(\frac{50}{10} \right)^{-\frac{1}{3}} = 0.585 \quad (1.9)$$

The above formula is based on the note in EN 1998-1, Sec. 2.1(4), where a formula is provided for the estimate of the importance factor when same probability of exceedance is

considered but different time period. This means that it is possible to calculate an importance factor for DLS assuming the identical probability of exceedance for ULS and DLS states, i.e. $P_{DLR} = P_{NCR} = 10\%$ and time period for ULS 50 years and 10 years for DLS.

Dynamic amplification factor beta0: this value β_0 is included for compatibility between all European national standards. For Eurocode this value is typically set to 2.5, but for instance for Romanian standards it can be 2.75 or 3.0 depending on the spectrum type. $\beta_0 = 2.50$ also for Romania acc. to CR6+P100 2013.

3.4 Materials

In this section user has to provide material information. User can add, edit, delete or import specific materials. This section is divided into several subsections for **Basic materials**, **Walls material**, **Lintels** and **RC Ring**. It is not necessary to define the materials before going to the definition of the structural geometry or DXF import as the program automatically asks the user to create the necessary material when it is required. So it is possible to skip the material definition and go directly to the **Topology** definition.

Basic materials: in this submenu basic material properties can be defined such as bricks, mortar, concrete and reinforcement. This basic material types are later used to define a composite material to be used for walls, lintels or for the reinforcement ring. The following basic material types are supported:

Bricks – bricks are selected from the catalogue of available Wienerberger products. The available choices depends on the selection of national localization.

Mortar – choice of available mortars depends on the selected national localization.

Concrete – Eurocode concrete classes are available

Reinforcement – this basic material is used to define basic reinforcement properties such as strength and elastic modulus.

Walls material: this submenu is used for the definition of a composite wall material, which usually consists of bricks, mortar and/or reinforcement. Each wall material is assumed to consist of the previously defined basic materials. The following wall materials are supported:

Unreinforced masonry – is used to represent wall consisting of bricks connected by mortar.

Reinforced masonry – is used to represent wall made of bricks with concrete infill and vertical or horizontal reinforcement.

Reinforced concrete – is used to model wall made of reinforced concrete, i.e. this wall material needs concrete and reinforcement basic materials.

Lintels: this submenu is used for the definition of lintels. The lintels are assumed to consist of concrete and reinforcement.

RC Ring: if your building includes a reinforced concrete ring at the level of ceilings, it will be necessary to define an appropriate composite material.

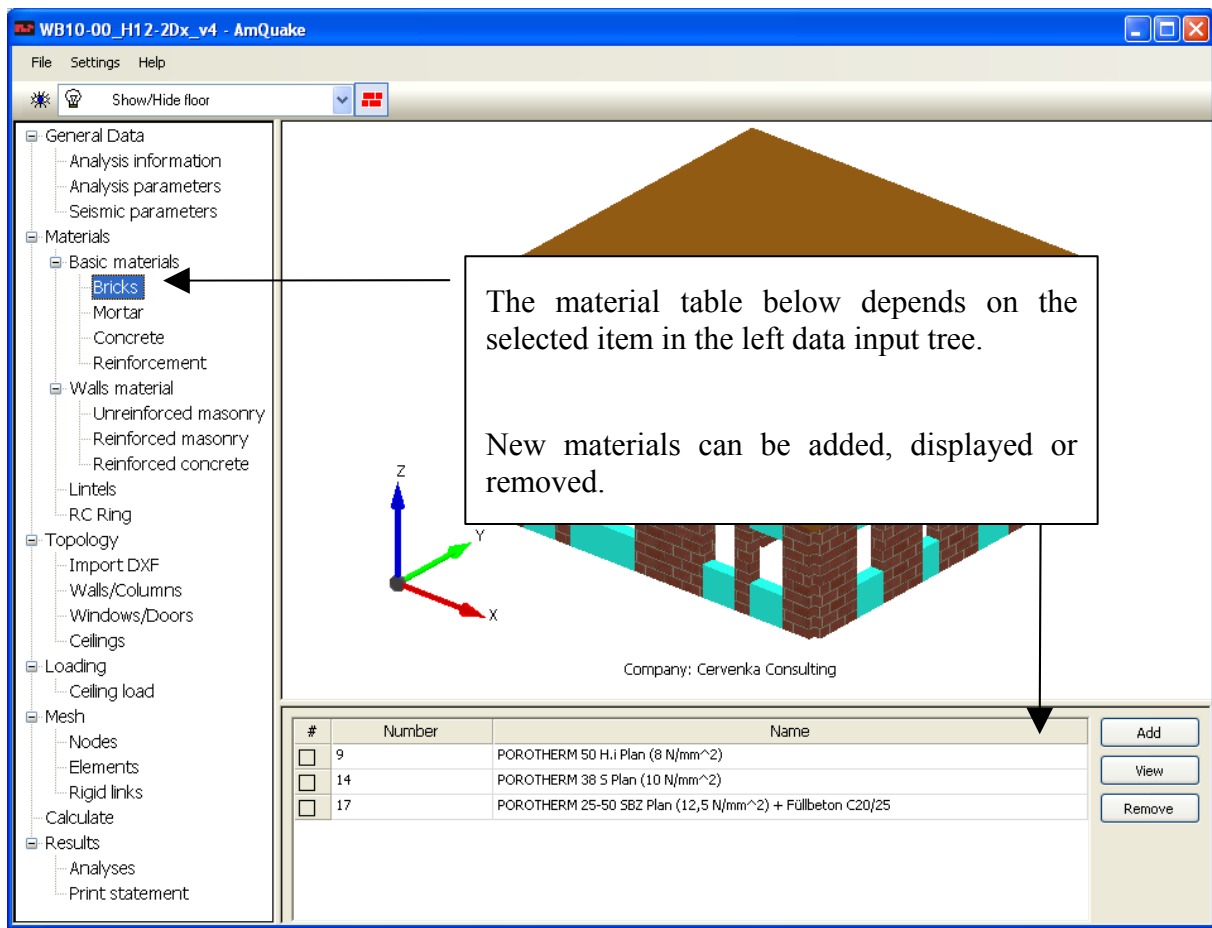


Figure 22: Material definition

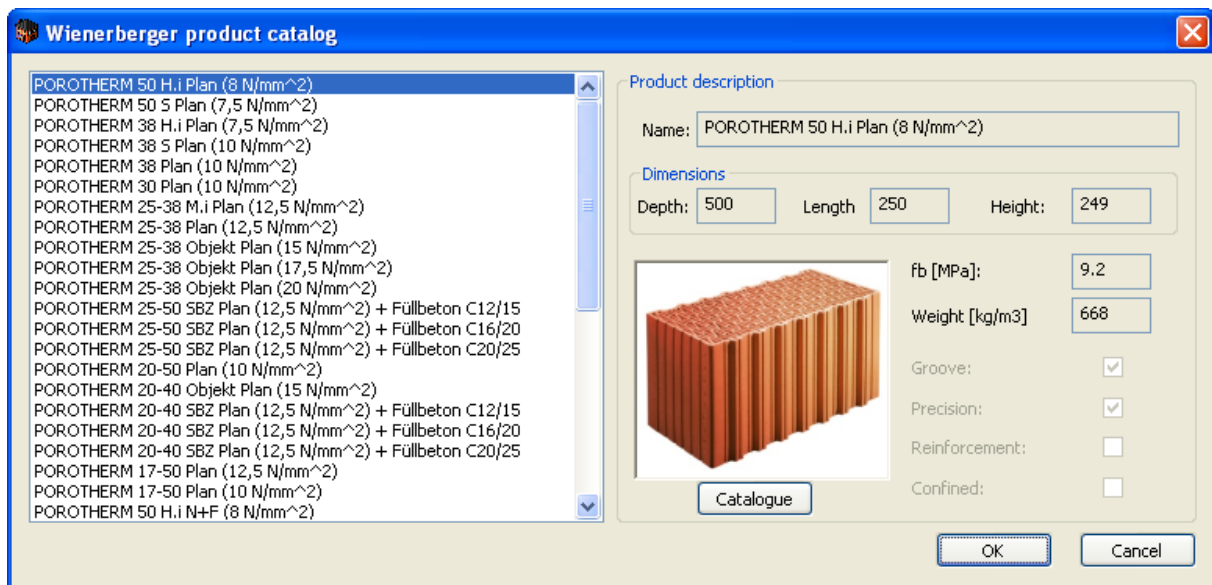


Figure 23: Catalogue of materials based on Wienerberger products, reinforcement and concrete types

3.4.1 Wall materials

The item **Wall material** in the data access tree is used to define a composite wall material. The **AmQuake** program recognizes the following wall materials: **Unreinforced masonry**, **Reinforced Masonry** and **Concrete**.

3.4.1.1 Unreinforced masonry

The unreinforced masonry is assumed to consist of brick units connected by mortar. The input dialog contains two property sheets.

Standard:

Here an appropriate brick is to be selected and a corresponding mortar. The program offers a selection of mortar types that are suitable for the given brick unit.

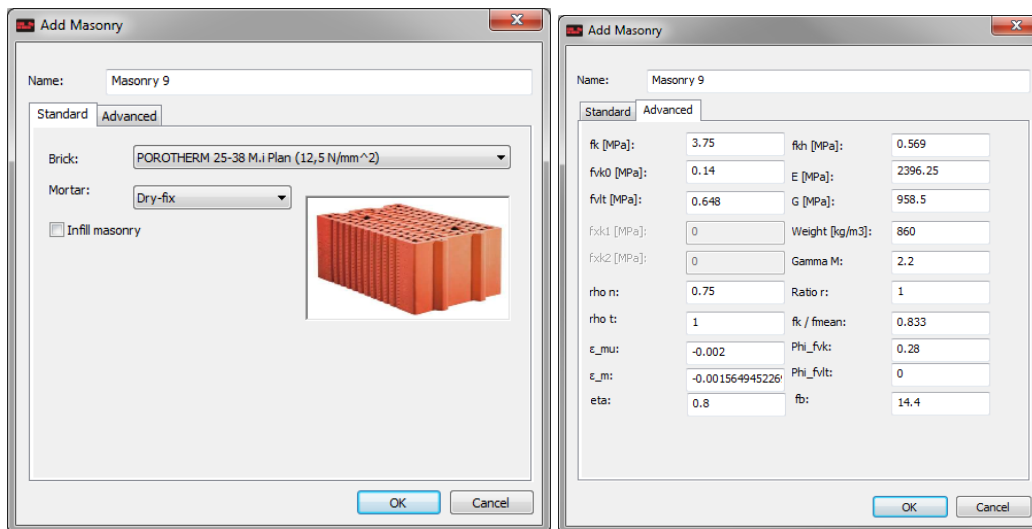


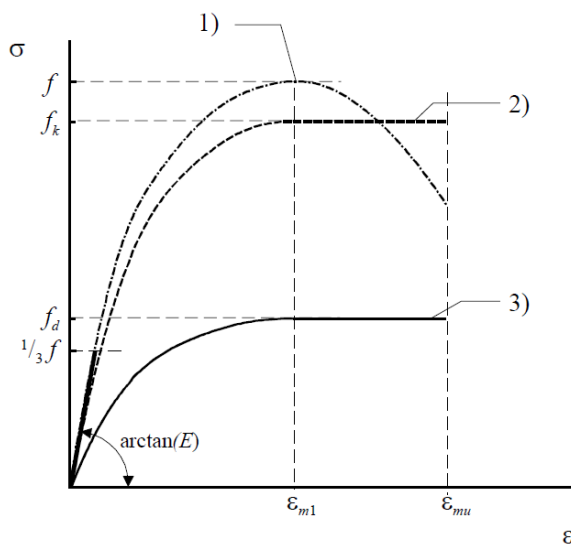
Figure 24: Unreinforced masonry wall material

Advanced:

The contents of this property sheet contain parameters of the composite masonry material. Based on the selected unit and mortar, the compressive strength, shear strength and other material parameters are generated using EN 1996-1-1 formulas. The individual entry fields can be modified manually. In this case it is recommended to select User defined brick type in **Standard** property sheet. The meaning of the individual parameters is defined below:

Name	= arbitrary string for easy identification of the specified material
f_k [MPa]	= characteristic compressive strength of masonry
f_{kh} [MPa]	= horizontal characteristic compressive strength of masonry
f_{vko} [MPa]	= characteristic initial shear strength
E [MPa]	= modulus of elasticity of masonry
G [MPa]	= shear modulus of masonry
Weight [kg/m³]	= specific masonry weight

γ_M	= partial factor of safety for masonry
r	= ratio of mortar thickness to the wall thickness
f_{vlt}	= maximal limit for shear strength
f_{xk1} [MPa]	= flexural strength, parallel to bed joints (not considered in the program)
f_{xk2} [MPa]	= flexural strength perpendicular to bed joints (not considered in the program)
ρ_n	= reduction factor for the wall effective height according to EN 1996-1-1, Sec. 5.5.1.2.
ρ_t	= reduction factor for the wall effective thickness according to EN 1996-1-1, Sec. 5.5.1.3.
ϵ_{mu}	= limiting compressive strain of masonry
ϵ_m	= compressive strain of masonry at compressive strength, equivalent to ϵ_{m1} from Figure 25.
f_k/f_{mean}	= ratio of characteristic to mean strength of the masonry wall. This value is used to convert the characteristic material properties to the mean one. This is required since mean parameters should be used in the pushover analysis as required by EN 1998-1, 4.3.3.4.1(4).
ϕ_{fvk}	= friction coefficient ϕ_{fvk} in Eq. (1.14). Typically the value of this coefficient is 0.4.
ϕ_{fvlt}	= friction coefficient ϕ_{fvlt} in Eq. (1.14). This value is used only in the Romanian code CR6+P100 otherwise the typical value is 0.0.
η	= coefficient to apply for F_D
f_b	= compressive strength of bricks



1) typical, 2) idealised, 3) design diagram

Figure 25: Stress-strain relationship of masonry

3.4.1.2 Reinforced masonry

The reinforced masonry is assumed to consist of brick units connected by mortar, with concrete infill and horizontal and vertical reinforcement. The input dialog (see Figure 30) in this case contains four property sheets.

Standard:

Name	= string for material identification
Bricks	= brick units suitable reinforced masonry
Mortar	= mortar types suitable for reinforced masonry brick units
Concrete	= concrete type to be used as an infill

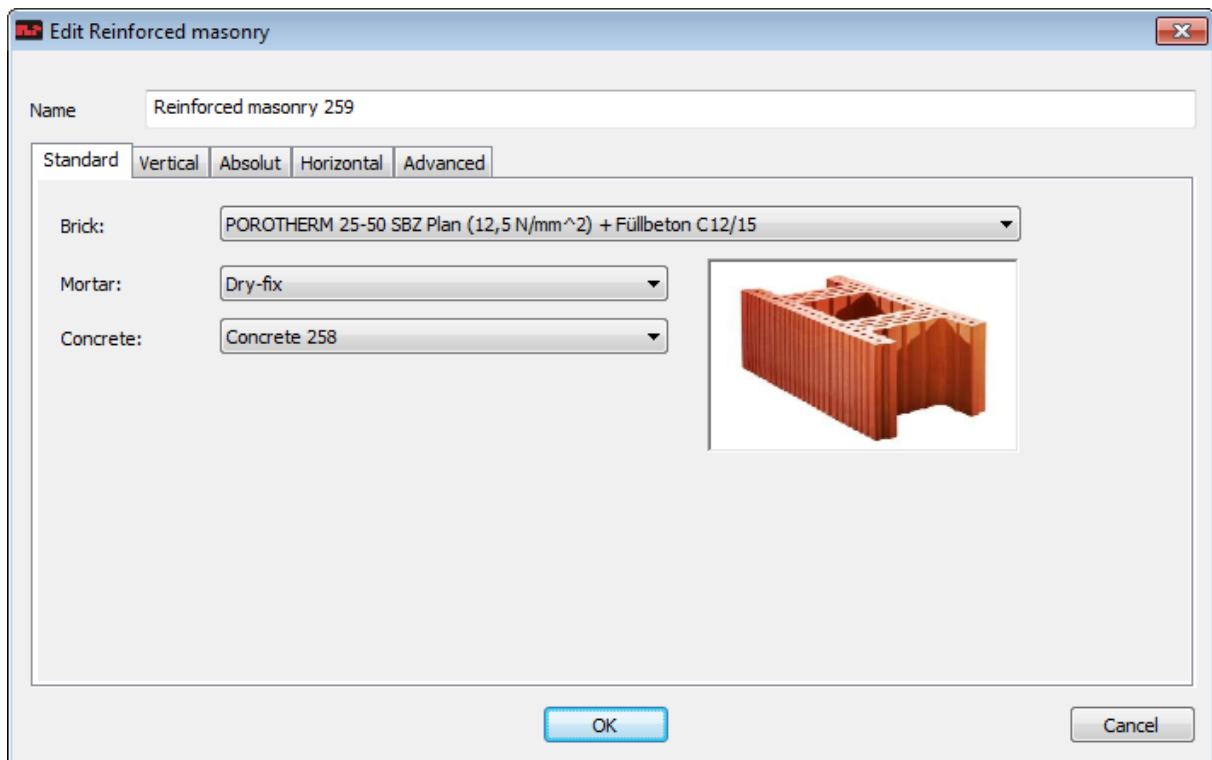


Figure 26: Dialog for reinforced masonry – standard sheet

Vertical reinforcement:

It is assumed that the reinforcement spacing along the wall edges may be different, therefore three values of reinforcement spacing are provided as well as the sizes of the left and right regions with the different reinforcement spacing (see Figure 30).

Left ring spacing	s_vl = spacing along the left wall edge
Left ring size	L_vl = size of the right reinforcement ring
Right ring spacing	s_vr = spacing along the right wall edge
Right ring size	L_vr = size of the right reinforcement ring
Default spacing	s_v = default spacing of the vertical reinforcement outside the left and right ring
Cover	= distance between reinforcement and external wall surfaces
Reinforcement	= reinforcement
Rows	= number of reinforcement rows/layers to be placed through the wall thickness

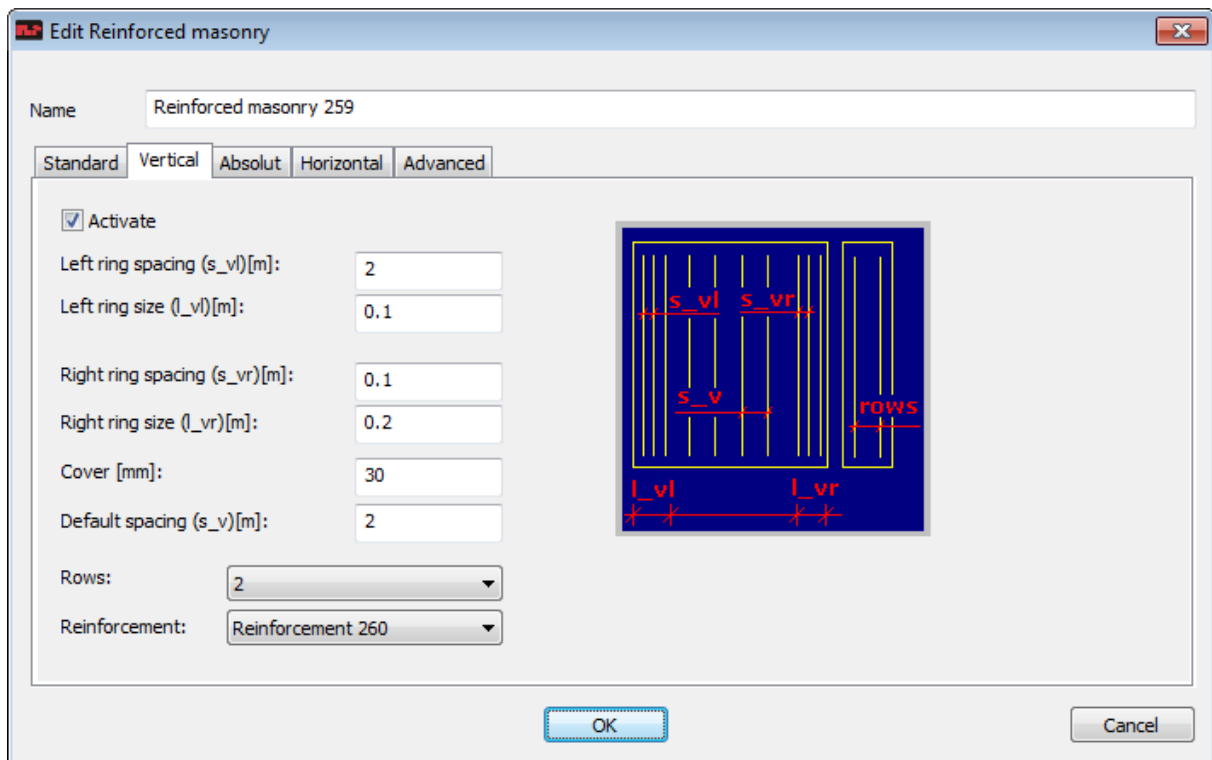



Figure 27: Dialog for reinforced masonry – vertical reinforcement definition

Absolute reinforcement:

Reinforcement is defined in absolute coordinates in local coordinate system of the wall. The local u axis is aligned along the wall length, i.e. local z-axis (see Figure 37). The local v axis is defined along the wall thickness. It has the same direction as the local y-axis (see Figure 37), but has a opposite orientation. The origin of the u-v coordinate system is at the bottom left corner of the wall when looking at the wall from its top to the bottom (i.e. along the local x-axis, see Figure 37). The buttons  can be used for quick generation of typical reinforcement configurations.

- u** = u coordinate of the centre of the reinforcement in the local u-v coordinate system
- v** = v coordinate of the centre of the reinforcement in the local u-v coordinate system
- c'** = reinforcement cover value used in the automatic generation of the cross-section
- Material** = reinforcement material. This material has to be defined previously in the definition of basic materials.
- Add** = add a new line into the reinforcement definition table
- Remove** = remove the selected line from the reinforcement table

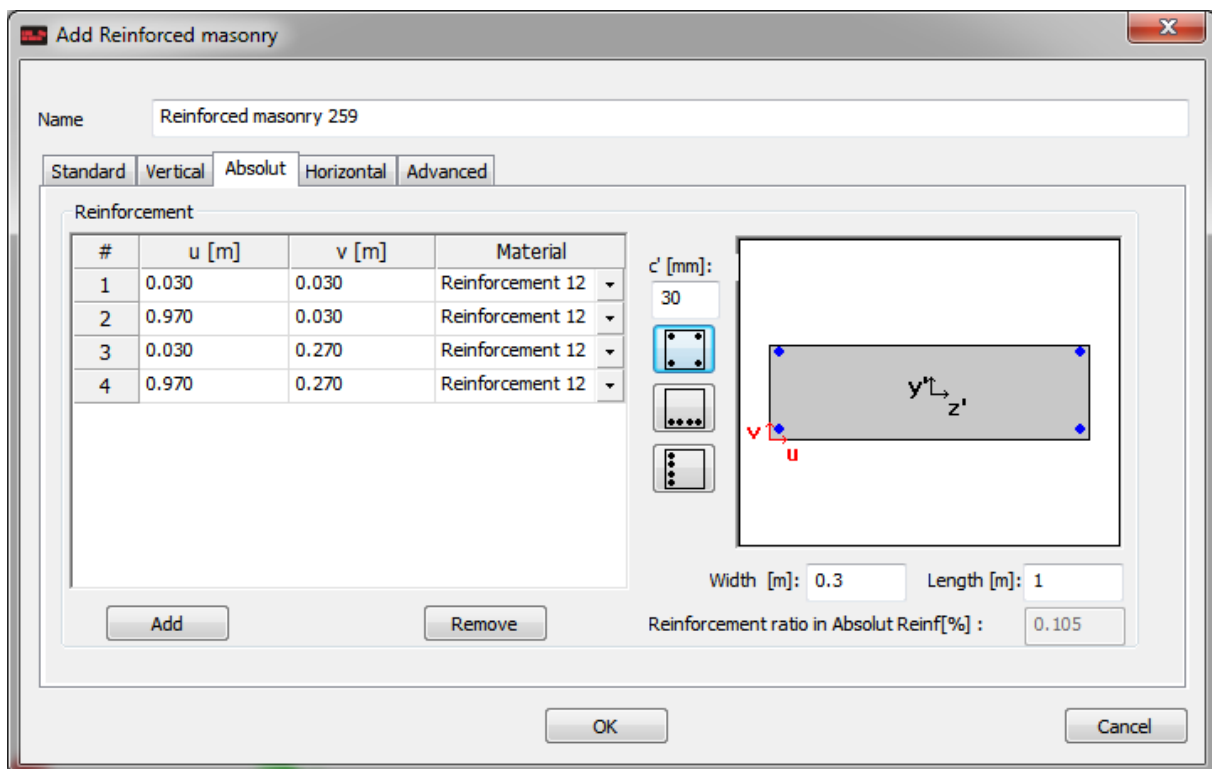


Figure 28: Dialog for reinforced masonry – reinforcement definition in absolut local wall coordinates

Horizontal reinforcement:

It is assumed that the horizontal reinforcement has a constant spacing in the vertical direction so the only parameter required is the spacing.

Stirrups = material for horizontal reinforcement

Spacing = spacing of horizontal reinforcement

Branches = how many branches are there for each layer of horizontal reinforcement

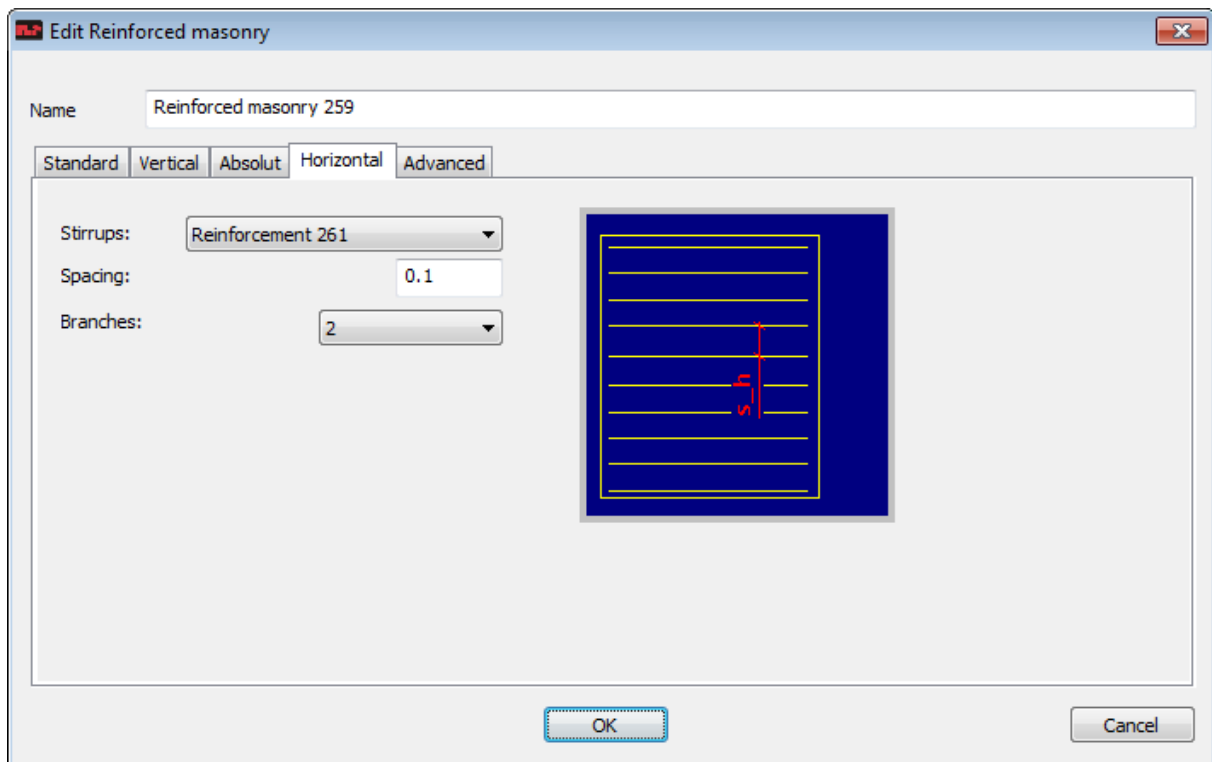


Figure 29: Dialog for reinforced masonry – definition of horizontal reinforcement

Advanced:

Advanced properties are generated automatically based on the selected masonry unit, mortar or concrete fill. In some national localization, it is possible to modify these properties manually.

f_k [MPa]	= characteristic compressive strength of masonry
f_{kh} [MPa]	= horizontal characteristic compressive strength of masonry
f_{vko} [MPa]	= characteristic initial shear strength
E [MPa]	= modulus of elasticity of masonry
G [MPa]	= shear modulus of masonry
Weight [kg/m ³]	= specific masonry weight
γ_M	= partial factor of safety
r	= ratio of mortar thickness to the wall thickness
f_{vl}	= maximal limit for shear strength
f_{xk1} [MPa]	= flexural strength, parallel to bed joints (not considered in the program)
f_{xk2} [MPa]	= flexural strength perpendicular to bed joints (not considered in the program)
ρ_n	= reduction factor for the wall effective height according to EN 1996-1-1, Sec. 5.5.1.2.

rho_t	= reduction factor for the wall effective thickness according to EN 1996-1-1, Sec. 5.5.1.3.
Eps_mu	= limiting compressive strain of masonry
Eps_m	= compressive strain of masonry at compressive strength, equivalent to ε_{m1} from Figure 25.
fk/fmean	= ratio of characteristic to mean strength of the masonry wall. This value is used to convert the characteristic material properties to the mean one. This is required since mean parameters should be used in the pushover analysis as required by EN 1998-1, 4.3.3.4.1(4).
Phi_fvk	= friction coefficient φ_{fvk} in Eq. (1.14). Typically the value of this coefficient is 0.4.
Phi_fvlt	= friction coefficient φ_{fvlt} in Eq. (1.14). This value is used only in the Romanian code CR6+P100 otherwise the typical value is 0.0.
eta	= coefficient to apply for F_D
fb	= compressive strength of bricks

Add Reinforced masonry

Name: Reinforced masonry 10

Standard Vertical Absolut Horizontal **Advanced**

fk [MPa]:	5.3	fk _h [MPa]:	0.731
fvk0 [MPa]:	0.14	E [MPa]:	3386.7
fvlt [MPa]:	0.27	G [MPa]:	1354.68
fvk1 [MPa]:	0	Weight [kg/m ³]:	1640
fvk2 [MPa]:	0	Gamma M:	2.2
rho n:	0.75	Ratio r:	1
rho t:	1	fk / fmean:	0.833
ε _{mu} :	-0.0035	Phi_fvk:	0.28
ε _m :	-0.001564945226	Phi_fvlt:	0
eta:	0.8	fb:	20

OK Cancel

Figure 30: Dialog for reinforced masonry – advanced properties

3.4.1.3 Reinforced concrete

This wall material is used to define section properties for reinforced concrete walls. The input fields in this dialog are divided into four property sheets

The screenshot shows a software dialog box titled "Edit Reinforced concrete". It has a standard Windows-style title bar with a close button. The dialog is divided into four tabs: "Standard", "Vertical", "Absolut", and "Horizontal". The "Standard" tab is currently selected. Inside the "Standard" tab, there is a "Name" field at the top containing the text "Reinforced concrete 263". Below the tabs, there are three sub-tabs: "Concrete:", "rho n:", and "rho t:". The "Concrete:" sub-tab is active, showing a dropdown menu with "Concrete 258" selected. Below this, the "rho n:" sub-tab shows a text input field with the value "0.75". The "rho t:" sub-tab shows a text input field with the value "1". At the bottom of the dialog, there are two buttons: "OK" and "Cancel".

Figure 31: Dialog for reinforced concrete material – standard sheet

Standard:

Concrete = select concrete type from the previously defined concrete materials. Alternatively, it is possible to select the item **Import new ...** to define a new concrete material. The newly defined concrete material is automatically added into the basic material concrete table.

rho_n = reduction factor for the wall effective height according to EN 1996-1-1, Sec. 5.5.1.2. This value is for simplicity assumed according to EN 1996-1-1. It is used for the calculation of additional moments due to the eccentricity of the normal force.

rho_t = reduction factor for the wall effective thickness according to EN 1996-1-1, Sec. 5.5.1.3. This value is for simplicity assumed according to EN 1996-1-1. It can be used for walls stiffened by piers, however in general in majority of cases it should be set to 1,0.

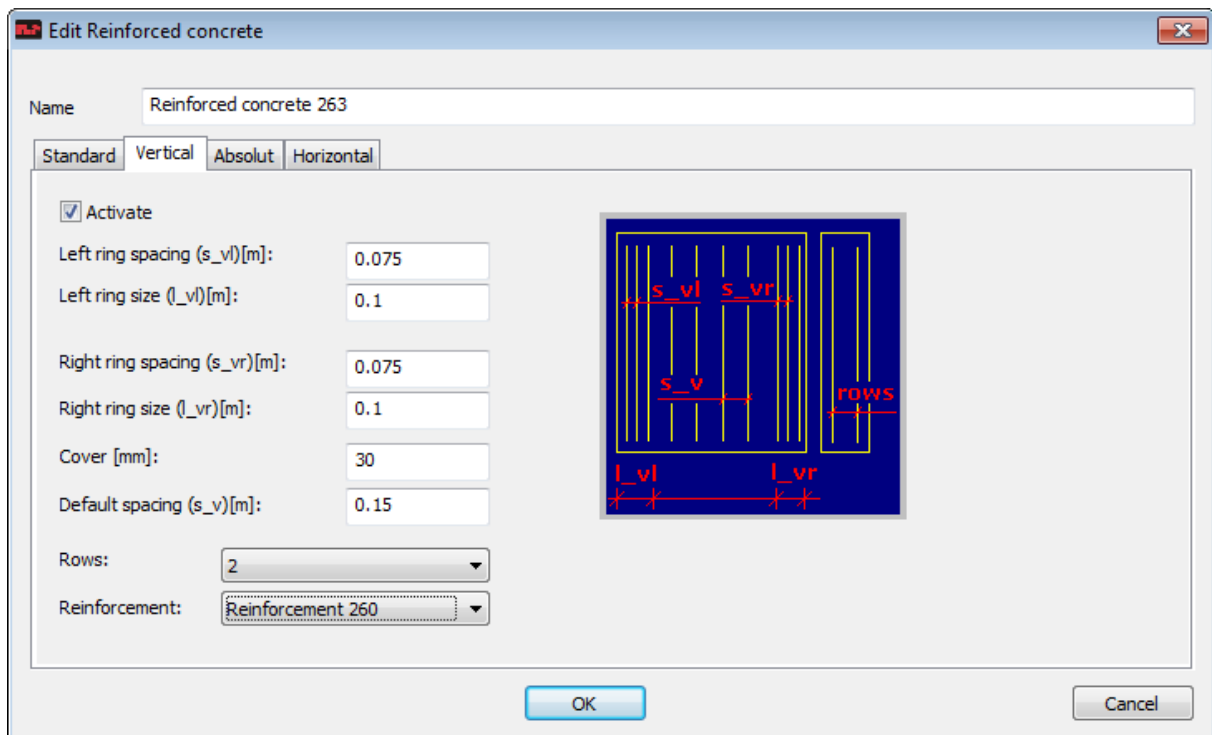


Figure 32: Dialog for reinforced concrete material – vertical reinforcement definition

Vertical reinforcement:

It is assumed that the reinforcement spacing along the wall edges may be different, therefore three values of reinforcement spacing are provided as well as the sizes of the left and right regions with the different reinforcement spacing (see Figure 30).

Left ring spacing	s_vl = spacing along the left wall edge
Left ring size	L_vl = size of the right reinforcement ring
Right ring spacing	s_vr = spacing along the right wall edge
Right ring size	L_vl = size of the right reinforcement ring
Default spacing	s_v = default spacing of the vertical reinforcement outside the left and right ring
Cover	= distance between reinforcement and external wall surfaces
Reinforcement	= reinforcement material
Rows	= number of reinforcement rows/layers to be placed through the wall thickness

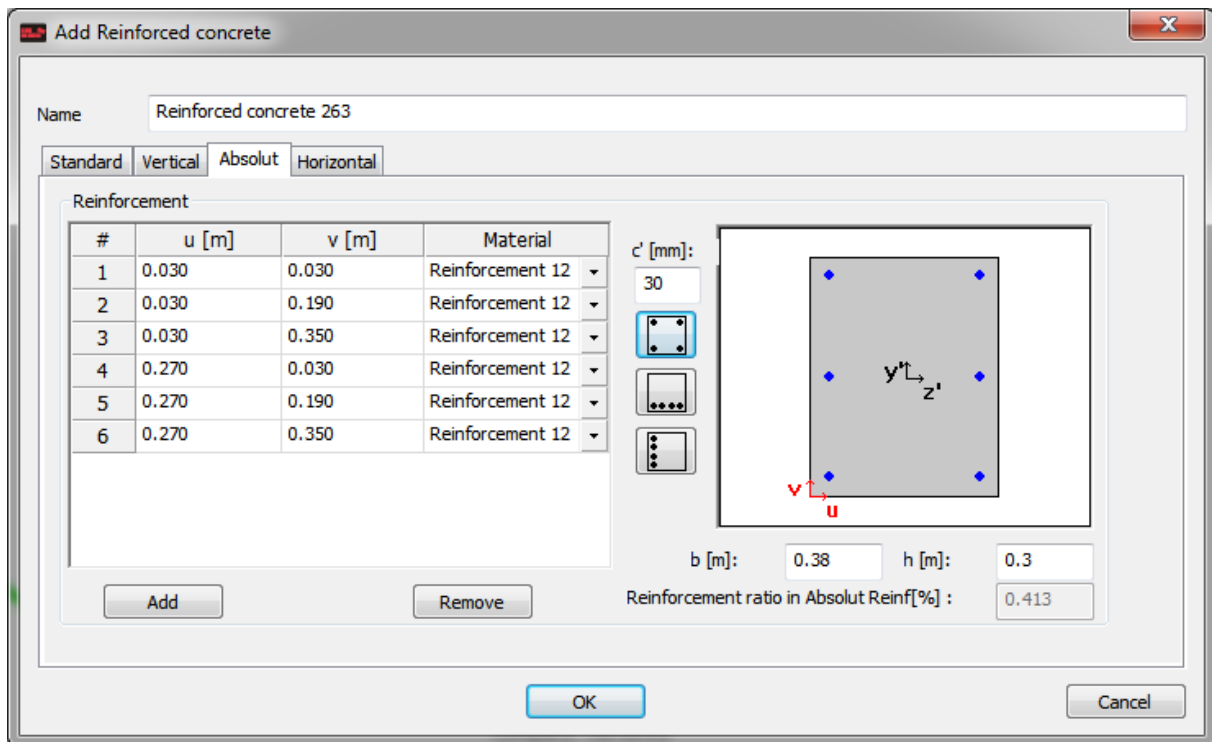



Figure 33: Dialog for reinforced concrete material – reinforcement definition in absolut local wall coordinates

Absolute reinforcement:

Reinforcement is defined in absolute coordinates in local coordinate system of the wall. The local u axis is aligned along the wall length, i.e. local z -axis (see Figure 37). The local v axis is defined along the wall thickness. It has the same direction as the local y -axis (see Figure 37), but has a opposite orientation. The origin of the u - v coordinate system is at the bottom left corner of the wall when looking at the wall from its top to the bottom (i.e. along the local x -axis, see Figure 37). The buttons  can be used for quick generation of typical reinforcement configurations.

u = u coordinate of the centre of the reinforcement in the local u - v coordinate system

v = v coordinate of the centre of the reinforcement in the local u - v coordinate system

c' = reinforcement cover value used in the automatic generation of the cross-section

Material = reinforcement material. This material has to be defined previously in the definition of basic materials.

Add = add a new line into the reinforcement definition table

Remove = remove the selected line from the reinforcement table

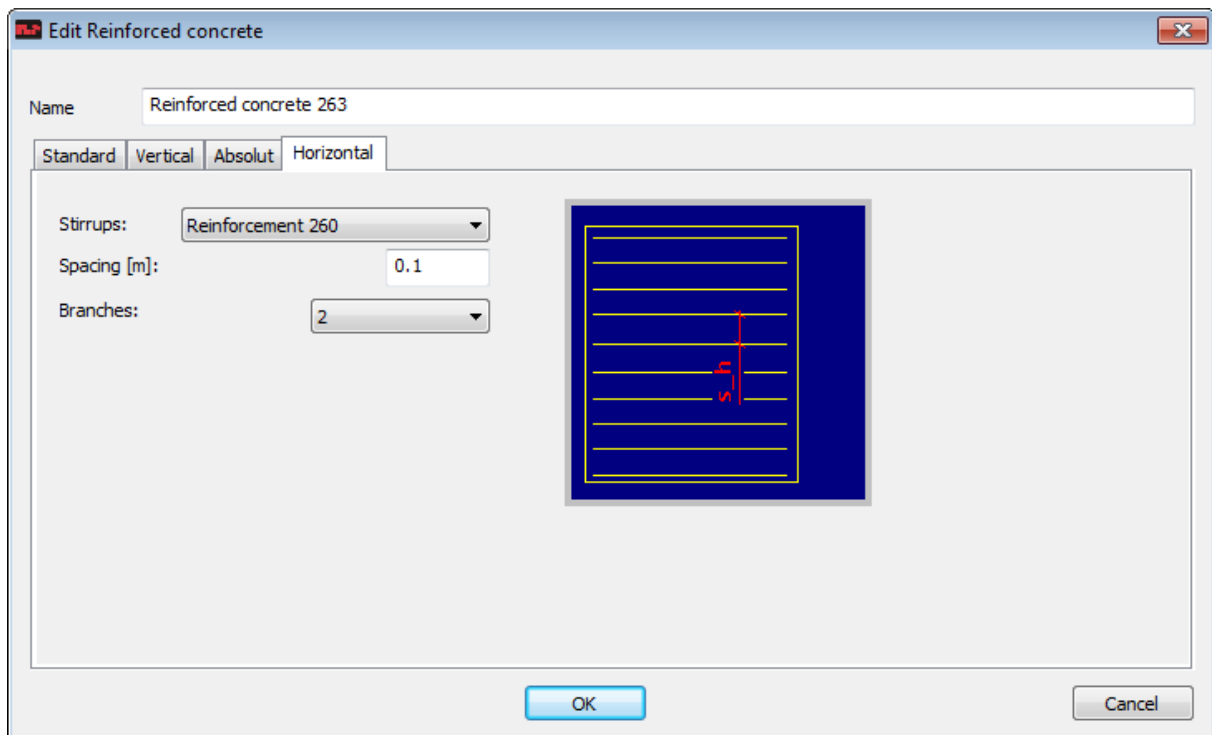


Figure 34: Dialog for reinforced concrete material – definition of horizontal reinforcement

Horizontal reinforcement:

It is assumed that the horizontal reinforcement has a constant spacing in the vertical direction so the only parameter required is the spacing.

Stirrups = material for horizontal reinforcement

Spacing = spacing of horizontal reinforcement

Branches = how many branches are there for each layer of horizontal reinforcement

3.4.1.4 Spandrel/lintel

This item is used for the definition of window lintels.

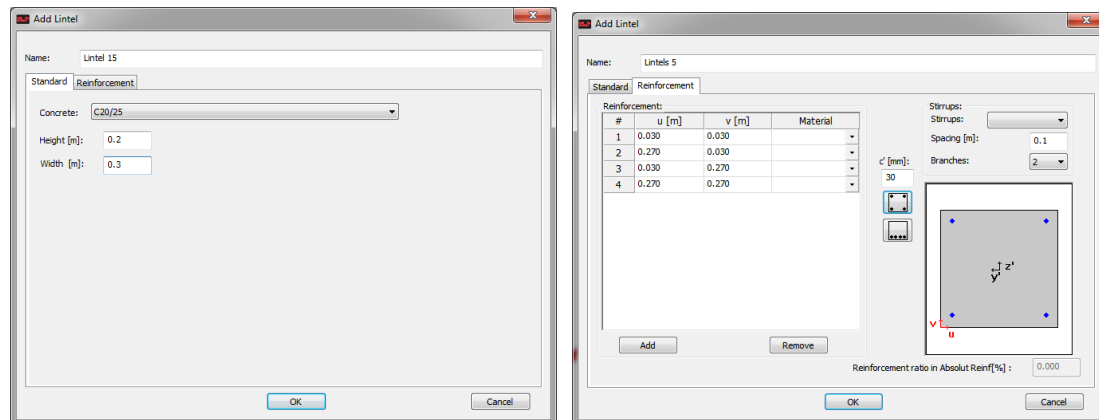


Figure 35: Spandrel/Lintel input dialog

The lintel dialog requires the definition of the following items:

Standard:

Name = lintel material name for easy identification

Concrete = previously defined concrete material can be selected here. If the necessary concrete material has not been defined yet, it is possible to select the item **Import new ...**, which will add a new concrete material to the appropriate table into the list of basic materials.

Height = height of the lintel

Width = width of the lintel


Reinforcement:

This property sheet of the lintel dialog is used for the definition of lintel reinforcement.

For each reinforcement it is possible to define its location in the cross-section of the lintel. The location is specified by two coordinates u , v , whose origin is in the left bottom corner.

Stirrups: = this input field defines the reinforcement material to be used for lintel stirrups. A previously defined reinforcement material can be selected here. If the necessary material has not been defined yet, it is possible to select the item **Import new ...**, which will add a new reinforcement material to the appropriate table into the list of basic materials.

Spacing = spacing of stirrups

The buttons  can be used to quickly generate typical reinforcement arrangements.

3.4.1.5 RC Ring

The definition of the RC ring is very similar to the lintel definition.

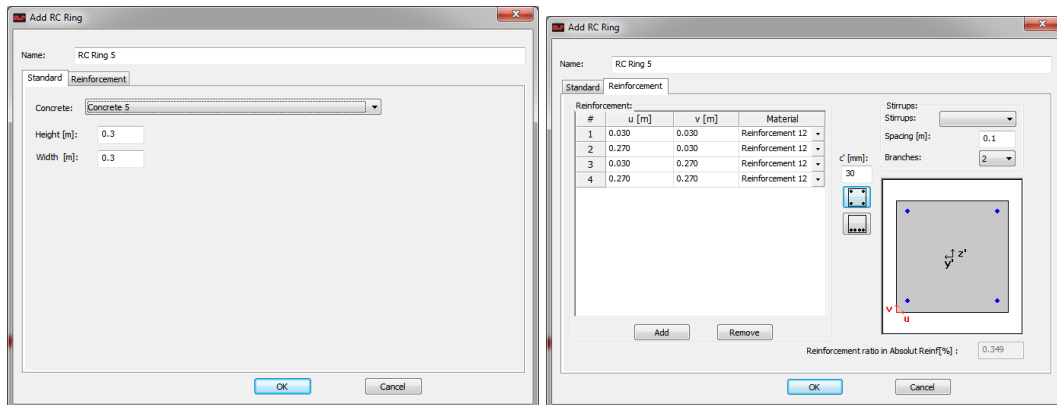


Figure 36: RC ring input dialog

The lintel dialog requires the definition of the following items:

Standard:

Name = lintel material name for easy identification

Concrete = previously defined concrete material can be selected here. If the necessary concrete material has not been defined yet, it is possible to select the item Import new ..., which will add a new concrete material to the appropriate table into the list of basic materials.

Height = height of the lintel

Width = width of the lintel


Reinforcement:

This property sheet of the lintel dialog is used for the definition of lintel reinforcement.

For each reinforcement it is possible to define its location in the cross-section of the lintel. The location is specified by two coordinates u , v , whose origin is in the left bottom corner.

Stirrups: = this input field defines the reinforcement material to be used for lintel stirrups. A previously defined reinforcement material can be selected here. If the necessary material has not been defined yet, it is possible to select the item **Import new ...**, which will add a new reinforcement material to the appropriate table into the list of basic materials.

Spacing s = spacing of stirrups

The buttons  can be used to quickly generate typical reinforcement arrangements.

More information about the modelling of reinforced concrete ring can be found in Section 4.7.

3.5 Topology

This section contains all the information about the geometry of the building to be analyzed. User can add manually elements such as walls and windows or can use the import dialog to import the data from the DXF file. The requirements on the content of this DXF file is described in Section 3.5.1.

The topology of the building is fully described by three types of geometric entities:

Walls/columns – this entity is used to model walls or columns in the analyzed building. Each wall is modelled as a beam element. In the case of extremely long walls, i.e. walls where the length is larger than the height, it is recommended to divide the wall into several wall next to each other.

Windows/doors – this geometric entity is used to model any kind of openings in the building, such as windows or doors. It is assumed that each window entity consists of a parapet, opening (i.e window) and lintel. These sub-entities are defined by their height. In case there is still some room left above the lintel, this remaining space it assumed to be built from the same material as the parapet.

Ceilings – they represent the ceiling slab above each floor. Each ceiling consists of several regions that are used to define the load distribution on to the walls below. It should be noted that the ceiling definition is used only for proper distribution of loads. **AmQuake** does not include the stiffness of the ceiling slab into the pushover analysis. Also the internal forces for the slab are not calculated and they are not checked with respect to the strength of individual slab sections. This should be verified by a separate calculation. **AmQuake** assumes that ceiling slab behaves as a rigid diaphragm in its plane. The bending stiffness of the slab is by default not considered, but optionally rigidity in bending can be also selected (see Section 3.6). For each ceiling it is possible to define a reinforced concrete ring, which will be automatically generated along the top faces of each wall and window from the floor level below. The bending stiffness of this r.c. ring is taken into account during the pushover analysis. Also all sectional forces of the ring are checked during the pushover analysis, and plastic hinges are inserted if necessary.

Walls, windows and ceilings can be defined manually by inserting and editing items in the corresponding table in the bottom right part of the program window. It is however, preferable to define the layout of each floor with the help of a dxf drawing.

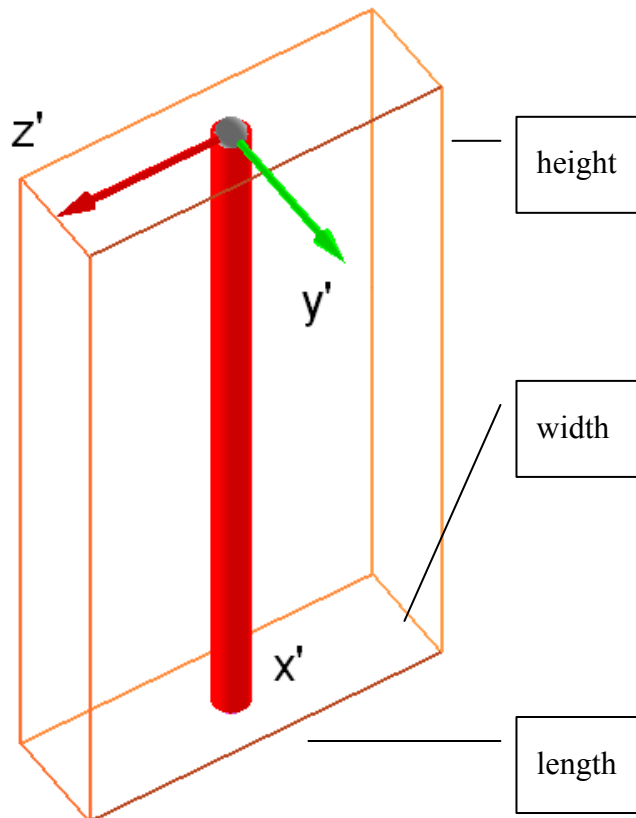


Figure 37: Local coordinate system of the wall and the wall finite element.

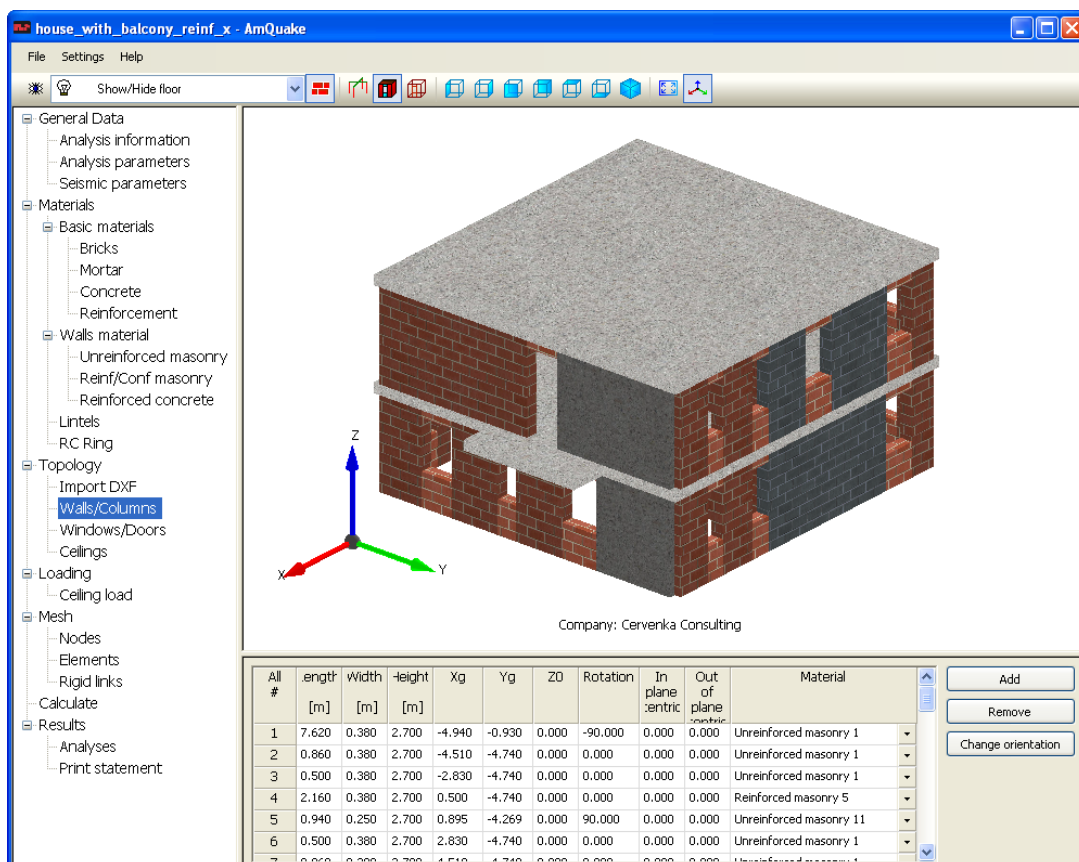


Figure 38: Description of various wall materials in AmQuake

3.5.1 Rules for DXF files

In order to import the layout of each floor from a DXF file, it is necessary to prepare this file according to specific rules, which are described in this section

Each floor layout has to be described by three separate layers. One layer should be created for walls, one for windows and one for the ceiling. These layers should use the same coordinate system, i.e. should be on top of each other. The names of the layer can be chosen arbitrarily by the user.

It is recommended to have all layers for all building levels in a single DXF file. In addition, it is advantageous to have all the layers with the same coordinates, i.e. on top of each other. If they are not on top of each other it is important to define a shift vector for each floor during the DXF import.

Walls and windows should be defined by rectangular boxes with all sides parallel.

A ceiling is defined by a set of lines. The lines should divide the ceiling area into several loading regions. The loading regions will be used to distribute the loading to the walls of the floor below. Program automatically checks, which walls are affected by each ceiling region. This means that each ceiling region applies the load to the walls that are found to be located below it.

The following rules apply to the definition of the ceiling areas:

- each loading area should contain the central point of at least one wall the belongs to the floor below the current ceiling.
- in case that the loading area contains several central points of various walls, the load is distributed in proportion to the wall cross-sectional areas.
- an opening in the ceiling can be defined by x,y coordinates of an arbitrary location. The program will search all the loading areas, and one, which contains this point will be considered as an opening. This means no loading will be considered for this ceiling region.

The windows/doors in AmQuake should observe the following rules:

- each window should have two neighbouring walls
- central points of window side edges should be near the central points of side edges of the neighbouring walls (see Figure 44). Figure 44(a, c) is a recommended way of modelling, but the new versions of AmQuake can handle also the case (b).
- side edges are edges that are perpendicular to the local Z' axis (see Figure 37)

The examples of layer for DXF file are in figures bellow.

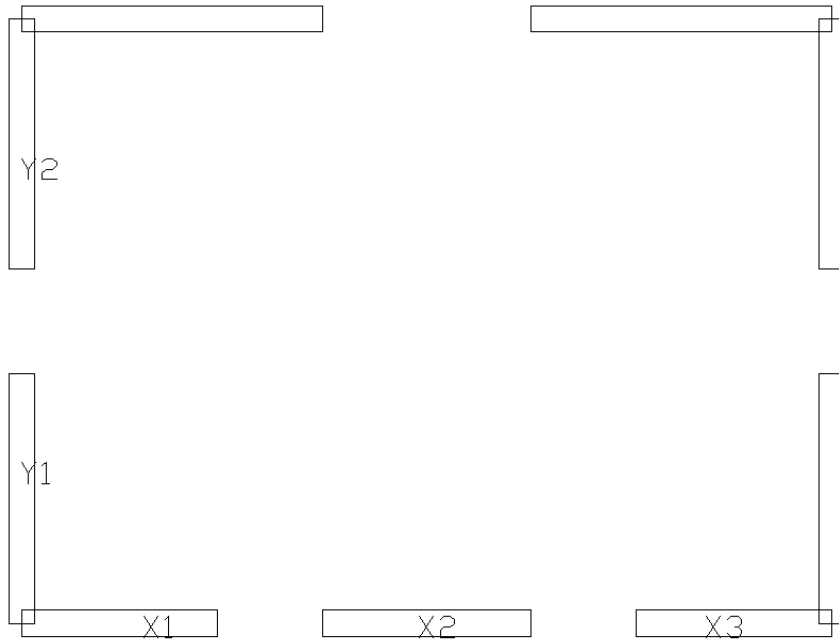


Figure 39: Example of a layer for walls

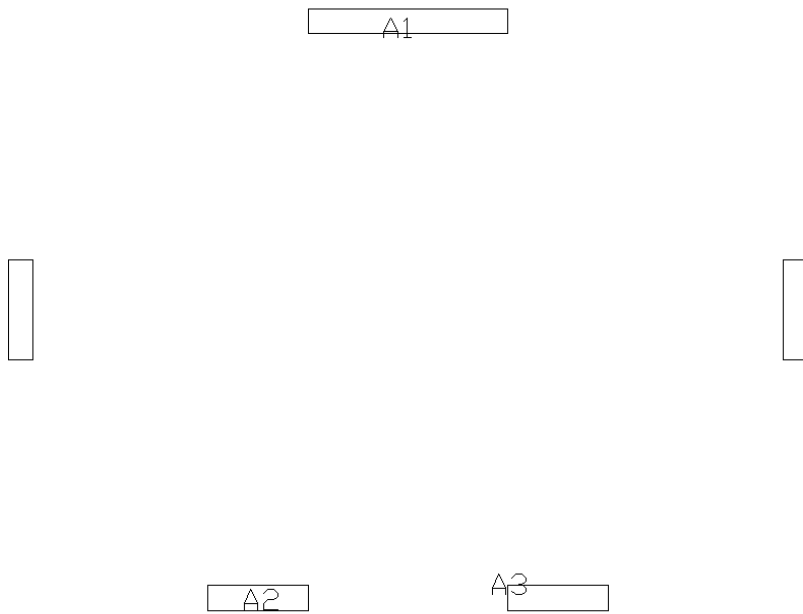


Figure 40: Example of a layer for windows

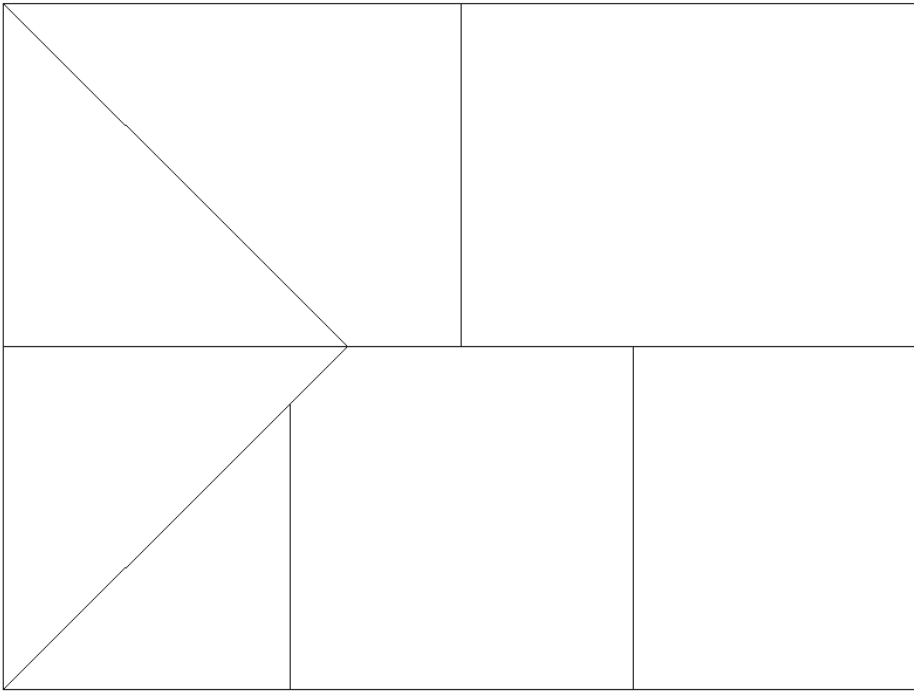


Figure 41: Example of a layer for a ceiling

Alternatively walls or windows can be defined also one or two diagonal lines. The walls defined only by one line must be parallel to the axes of x or y (their rotation against the x and y axes must be zero). The wall defined by two diagonal lines can be in any direction. Examples of simple definitions are in the figure bellow.

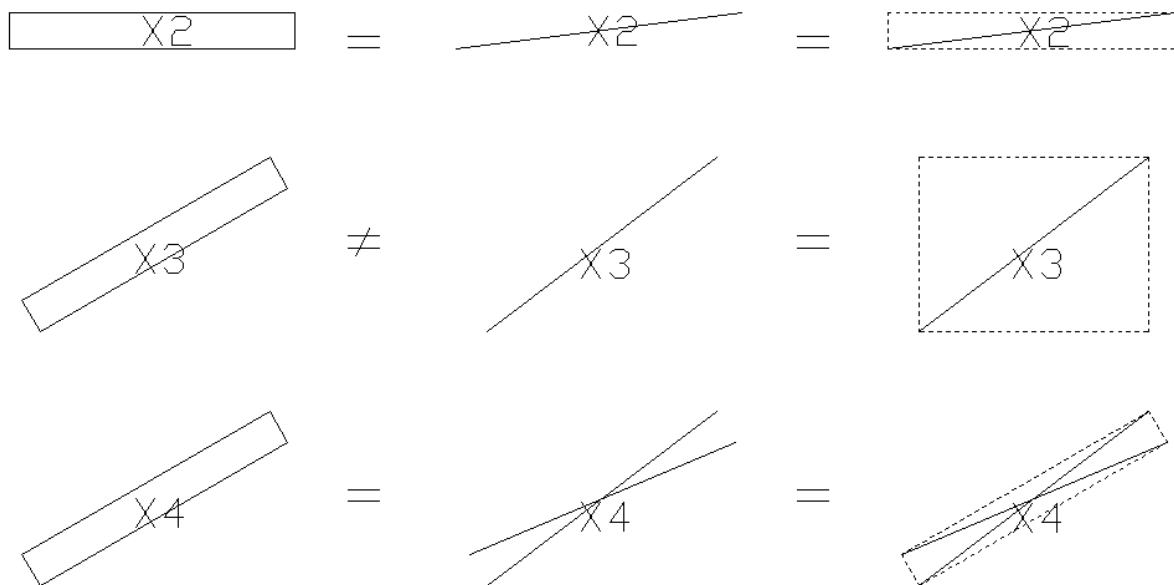


Figure 42: Example of a wall or window definition in a DXF file

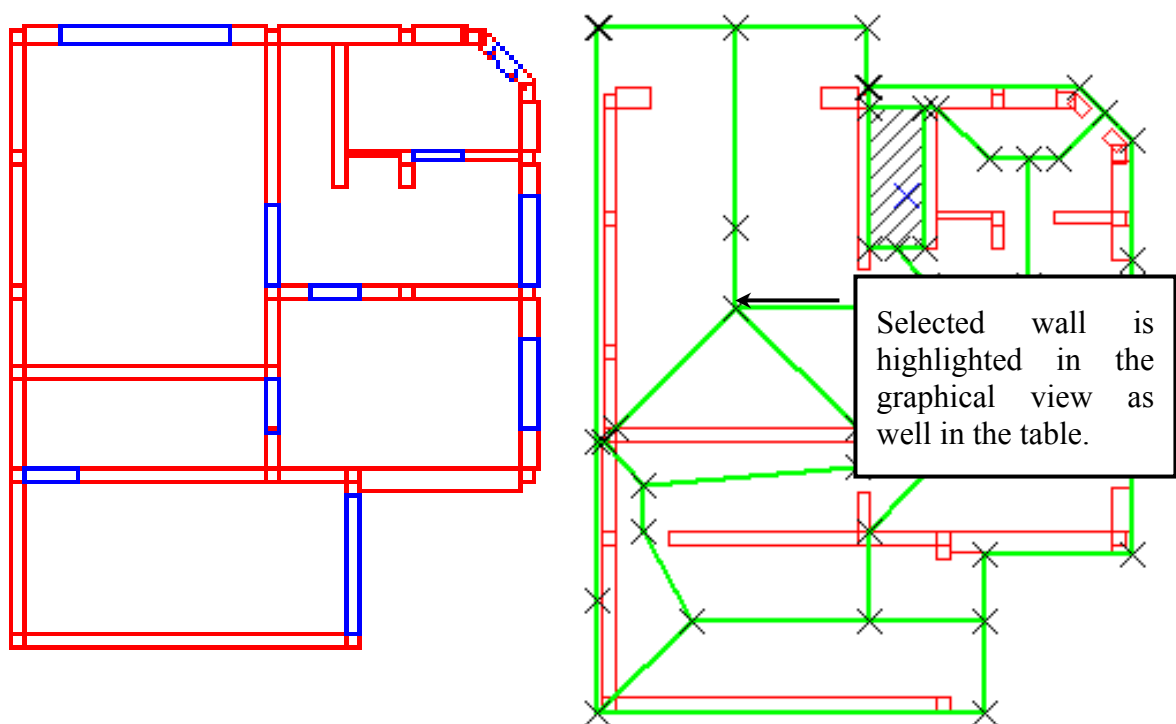


Figure 43: Example of wall, windows and ceiling definition in a DXF file.

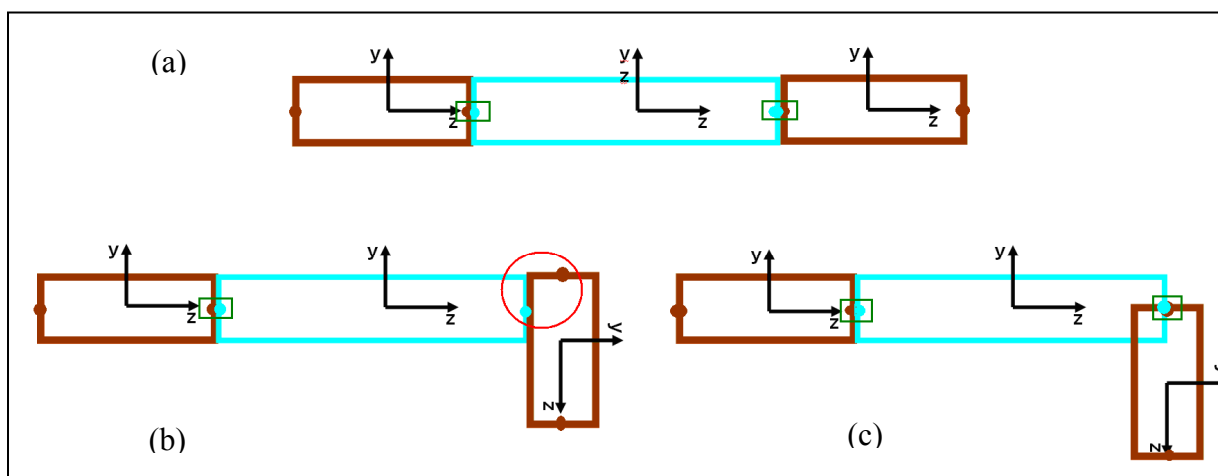


Figure 44: Possible definition of connection between windows and walls

3.5.2 Import DXF

Clicking on **Import DXF** button user opens the import dialog. In this dialog a DXF file can be selected, and all the topological entities are created automatically based on the contents of this file.

It is recommended first to Load DXF file and then a user continues with the definition of walls, windows and ceilings. It should be noted that the content of the dialog changes depending on the selected topology. A user should first select the topology and then choose the layer from the dxf file, which contains the definition of the selected geometrical entities.

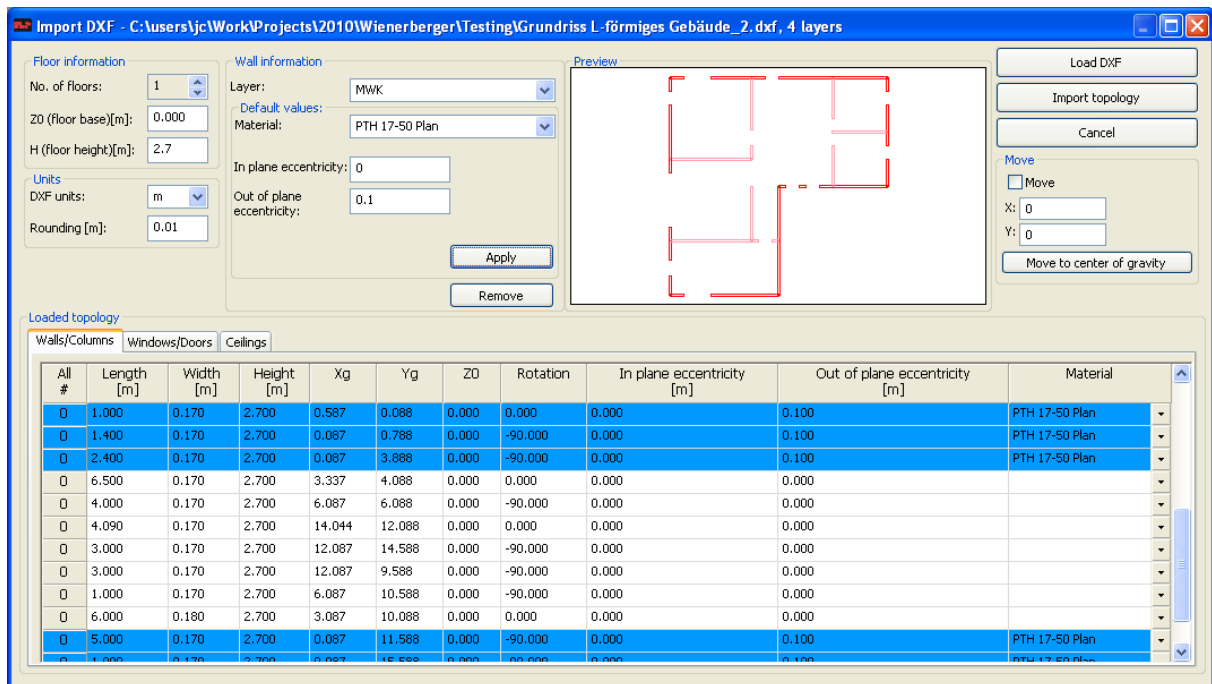


Figure 45: Import DXF - walls

No. of floors = number of floors to be generated based on the currently selected dxf file.

Z0 (floor base) = elevation of the floor to be generated. If the number of floors is greater than 1, the subsequent floors are automatically placed above the current one. The elevation of an each subsequent floor is calculated using the following formula:

$$Z_{n+1} = Z_n + H + \text{Ceiling height of the highest floor (m)}$$

H (floor height) = the height of the new floor. This corresponds to the wall height. The height of the ceiling is not part of H.

Units:

DXF Units = units that are used in the DXF file. The dimensions in the DXF will be converted into the default units of the program AmQuake.

Rounding = this defines the precision for rounding the dimensions provided in the DXF file.

Move = when this check box is activated, the walls, windows and ceilings are shifted to a new position. First an appropriate shift vector should be defined in the X, Y coordinate fields below. The translation into the new position is performed when the Move checkbox is selected.

This shift must be used if the layers for each floor do not use the same coordinate system, i.e. do not have the same origin.

Move to centre of gravity

This button works together with the Move button. It automatically generates values of the shift vector such that the origin of the coordinate system is in the centre of gravity based on the walls and windows that are currently listed in the tables of the import dialog.

Wall information

In order to define walls based on the selected DXF file, it is necessary to activate the **Walls/Columns** table in the bottom part of the dialog.

Layer = the selection of the layer containing the wall definition

Material = in this field, it is possible to select the wall material from the table previously created materials. If a suitable material does not exist, it is possible to define it by selecting one of the available **Import** functions.

In plane eccentricity = this defines the in plane eccentricity.

Out of plane eccentricity = this defines the out of plane eccentricity.

Apply = this button applies the currently specified material and eccentricities on the selected walls in the table at the bottom of the import dialog.

Remove = this button removes the selected walls from the import table. This means that the walls will not be imported into the current model.

The program automatically generates new walls based on the information provided in the loaded DXF file and based on the selected layer. The generated walls are listed in the table at the bottom part of the dialog.

Walls/Columns:

= wall identification

Length = length of the wall

Width = width of the wall

Height = height of the wall. It should be identical to the floor height above.

Xg = x-coordinate of the centre of the bottom face of the wall

Yg = y-coordinate of the centre of the bottom face of the wall

Z0 = z-coordinate of the centre of the bottom face of the wall. This value should be identical to the floor base above

Rotation = rotation of the wall in the plane of the floor

Material = material, which is to be used for this wall element

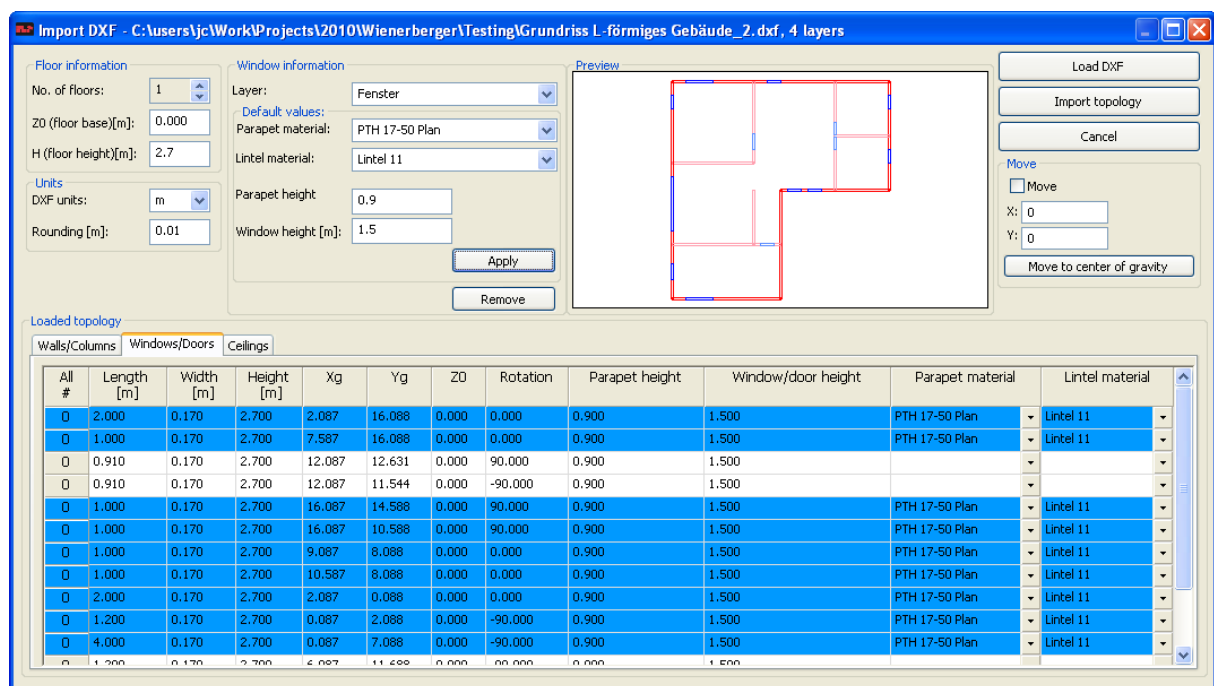


Figure 46: Import DXF - windows

Windows/Doors:

Layer = the selection of the layer containing the window definition

Parapet material = in this field, it is possible to select the wall material from the table previously created materials. If a suitable material does not exist, it is possible to define it by selecting one of the available **Import** functions.

Lintel material = in this field, it is possible to select the wall material from the table previously created materials. If a suitable material does not

exist, it is possible to define it by selecting one of the available **Import** functions.

Parapet height	= height of the parapet below the window
Window height	= height of the window. The lintel will be placed directly above the window. If there is space remaining above the lintel, it is assumed that this remaining space will be filled with the same material as for the parapet.
Apply	= this button applies the currently specified material and eccentricities on the selected walls in the table at the bottom of the import dialog.
Remove	= this button removes the selected walls from the import table. This means that the walls will not be imported into the current model.

The program automatically generates new windows based on the information provided in the loaded DXF file and based on the selected layer. The generated windows are listed in the table at the bottom part of the dialog.

Windows/Doors:

#	= window identification
Length	= length of the window
Width	= width of the window
Height	= total height of the window. It should be identical to the floor height above.
Xg	= x-coordinate of the centre of the bottom face of the window
Yg	= y-coordinate of the centre of the bottom face of the window
Z0	= z-coordinate of the centre of the bottom face of the window. This value should be identical to the floor base above
Rotation	= rotation of the window in the plane of the floor
Parapet height	= height of the parapet below the window
Window height	= height of the window
Parapet material	= material, which is to be used for this parapet element below the window. This material is also used for any remaining space above the lintel.
Lintel material	= material for the lintel above the window

#	Name	Area [m2]	Height [m]	Number of geometry points	Number of loading areas	Z	RC Ring
1		197.6306	0.3000	6	1	11.4600	

Figure 47: Import DXF - ceiling

Ceiling information:

Layer = the selection of the layer containing the definition of ceiling
 Ceiling height = height/thickness of the ceiling

The program automatically generates the ceiling geometry based on the information provided in the loaded DXF file and based on the selected layer. The generated definition is listed in the table at the bottom part of the dialog.

The ceiling definition is used only for proper distribution of loads. **AmQuake** does not include the stiffness of the ceiling slab into the pushover analysis. Also the internal forces for the slab are not calculated and they are not checked with respect to the strength of individual slab sections. This should be verified by a separate calculation. **AmQuake** assumes that ceiling slab behaves as a rigid diaphragm in its plane. The bending stiffness of the slab is by default not considered, but optionally rigidity in bending can be also selected (see Section 3.6).

Ceilings:

Name = Ceiling identification
Area = Surface area of the ceiling
Height = height, i.e. thickness of the ceiling structure
Number of geometry points = number of geometrical points defining the ceiling geometry and loading areas

Number of loading areas	= number of loading areas/regions that were identified in DXF file. It is possible to double click in this field to edit the loading area definition. This opens a dialog, where new loading regions can be manually inserted or an invalid definition can be corrected. See Section 3.5.5 for more details.
Z	= Z-coordinate of the bottom surface of the ceiling slab
RC Ring	= RC Ring material, which needs to be defined previously in the Materials definition. RC ring can be also added later after by editing the ceiling definition.

For each ceiling it is possible to define a reinforced concrete ring, which will be automatically generated along the top faces of each wall and window from the floor level below. The bending stiffness of this r.c. ring is taken into account during the pushover analysis. Also all sectional forces of the ring are checked during the pushover analysis, and plastic hinges are inserted if necessary. See also Section 4.7 for more information about correct r.c. ring definition.

3.5.3 Topology – walls / columns

This item in the data entry tree, opens a table containing a list of walls. New walls can be manually added here or deleted. It should be noted that the program does not differentiate between walls or columns. A column is just a wall with the ratio of thickness to length that is closer to one. A particular wall can be selected either by clicking at the wall directly in the window containing the view of the structure or by clicking the small box at the left side of the wall table. Each wall is defined by the following set of data:

Number	= automatically created id of the wall
Length	= wall length
Width	= width, i.e. thickness
Height	= wall height
Xg	= x-coordinate of the bottom of the wall
Yg	= y-coordinate of the bottom of the wall
Z0	= z-coordinate, i.e. elevation at the bottom of the wall. These coordinates define the centre of the bottom edge of the wall
Rotation	= rotation of the wall in the plan view of the building in degrees
In plane eccentricity	= this defines the in plane eccentricity.
Out of plane eccentricity	= this defines the out of plane eccentricity.
Material	= material to be used for the selected wall.

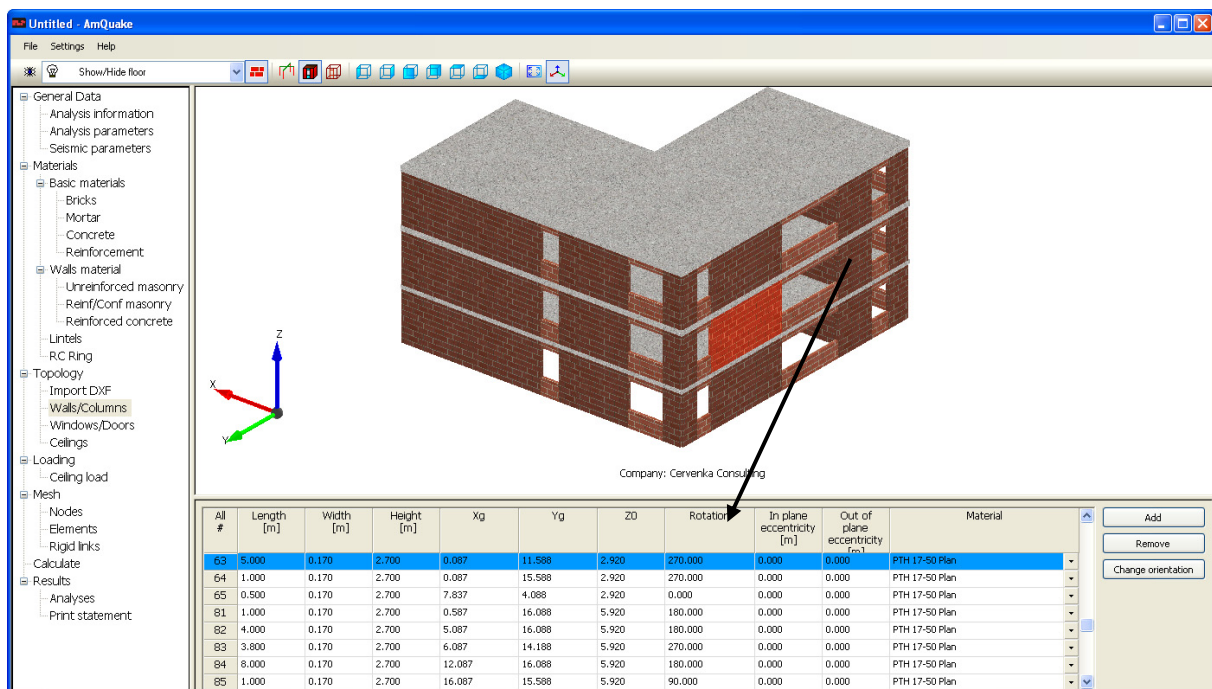


Figure 48: Topology – Walls / Columns

3.5.4 Topology – windows / doors

This item in the data entry tree, opens a table containing a list of windows/doors. A new window can be manually added here or deleted. A particular opening can be selected either by clicking at the entity directly in the window containing the view of the structure or by clicking the small box at the left side of the table. Each window/door is defined by the following set of data:

Number	= automatically created id of the window/doors
Length	= window length
Width	= width, i.e. thickness
Height	= window/door height
Xg	= x-coordinate of the centre of the bottom of the window
Yg	= y-coordinate of the centre of the bottom of the window
Z0	= z-coordinate of the centre of the bottom of the window
Rotation	= rotation of the window in the plan view of the building in degrees
Parapet height	= height of the window parapet. Can be set to zero, if no parapets are to be included
Parapet material	= material for the parapet
Lintel material	= material for the lintel

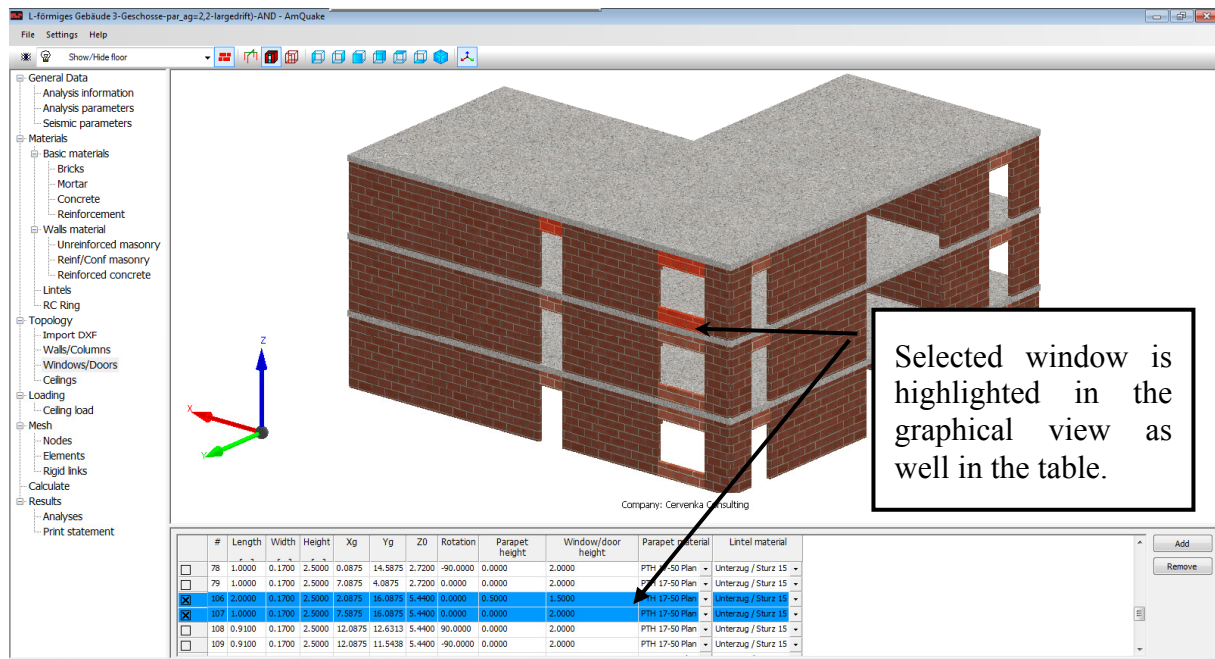


Figure 49: Topology – Windows / Doors

3.5.5 Topology – ceilings

This item in the data entry tree, opens a table containing a list of ceilings. A new ceiling can be manually added here or deleted. A particular selected can be selected either by clicking at the entity directly in the window containing the view of the structure or by clicking the small box at the left side of the table. Each ceiling is defined by the following set of data:

- Name** = ceiling identification
- Area** = ceiling area
- Height** = height, i.e. thickness of the ceiling structures
- Number of geometry points** = number of points defining the ceiling. This field can be double-clicked, which opens a dialog window when the individual points can be modified or new geometry points can be included.
- Number of loading areas** = number of loading areas defining the distribution of ceiling loads to the individual walls. This field can be double-clicked, which opens a dialog window when the individual points as well as loading areas can be modified or new ones can be included. (See Figure 51)
- Z** = ceiling elevation
- RC Ring** = material for the reinforced concrete ring. If this material is specified elements will be included into the numerical model to simulate a reinforced concrete ring.

For each ceiling it is possible to define a reinforced concrete ring, which will be automatically generated along the top faces of each wall and window from the floor level below. The bending stiffness of this r.c. ring is taken into account during the pushover analysis. Also all

sectional forces of the ring are checked during the pushover analysis, and plastic hinges are inserted if necessary. See also Section 4.7 for more information about correct r.c. ring definition.

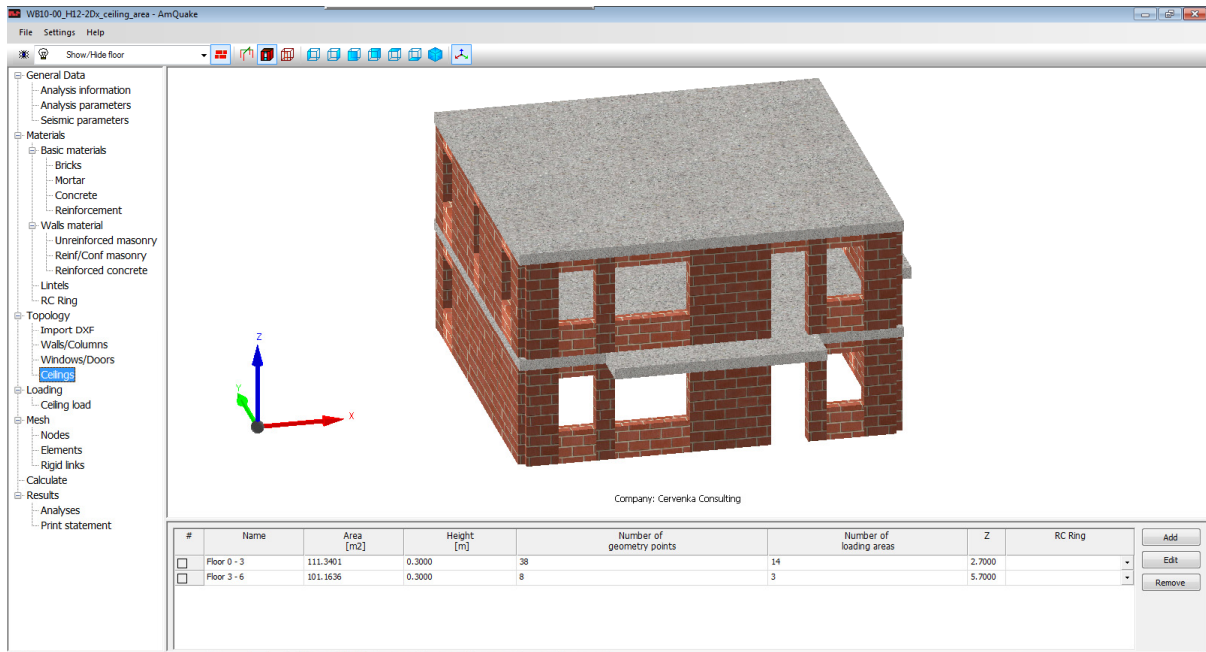


Figure 50: Topology – Ceilings

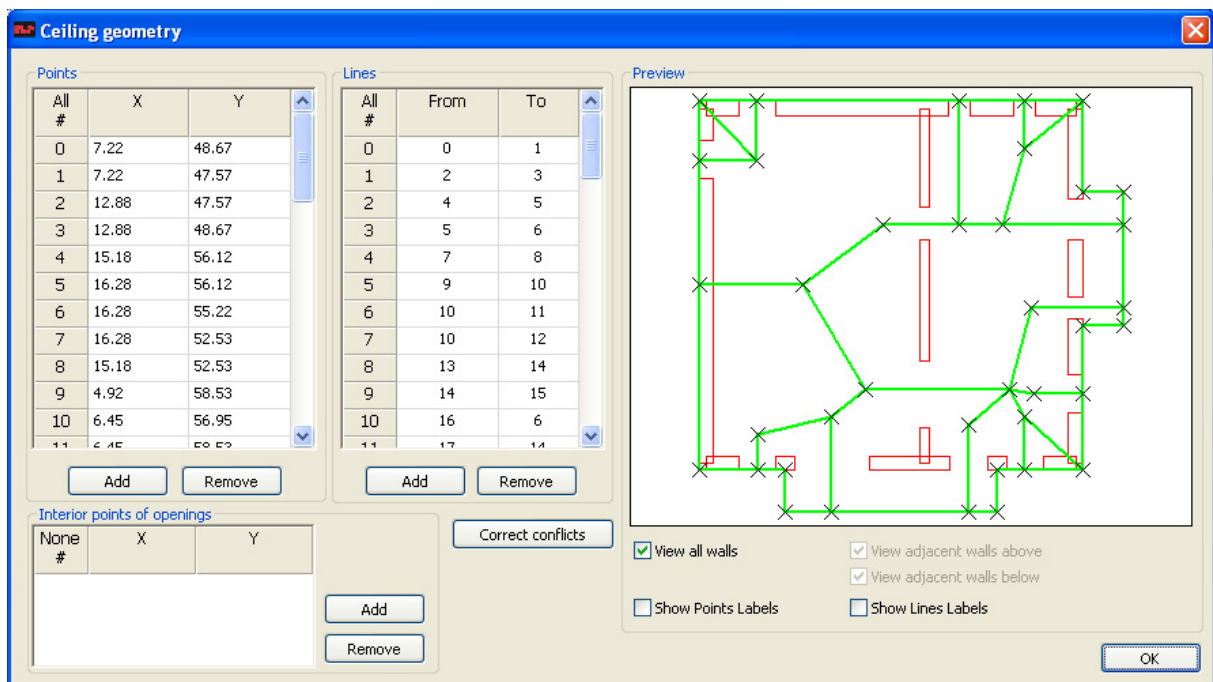


Figure 51: Ceiling Geometry and Loading Areas

Correct conflicts = this button can be used to automatically correct invalid intersections of lines

Interior points of openings = here x, y, coordinates can be added. Each region, which contain at least one of these points is interpreted as an opening, i.e. its load will

not be considered in the analysis. The point can be anywhere inside the region, which should be treated as an opening.

In the definition of loading areas, it is important to pay close attention to their definition, especially when large loading area is defined for a single wall. If large loading area is assigned to a single wall with small cross-section, the wall may collapse early on during the analysis. It should be carefully evaluated, if such a load distribution is realistic.

It is possible to define a single load distribution area for several walls. In this case the load is distributed according to the cross-sectional area of the associated walls.

Loading

In this section of the data input tree, the user defines the vertical loads acting on the structure.

3.5.6 Ceiling load

This item in the input data tree activates a table, where the loading information for each ceiling can be specified or modified.

Floor	= floor/ceiling identification
Dead load	= self weight of the ceiling structure
Permanent load	= weight of the additional permanent floor elements
Pushover Gamma_Gp	= permanent load G multiplier $\gamma_{G,p}$ for pushover analysis
Static Gamma_Gs	= permanent load G multiplier $\gamma_{G,s}$ for static analysis
Life load	= life load Q_L acting on the ceiling
Pushover Phi_L	= φ_L coefficient for life load Q_L and pushover analysis
Pushover Psi_2L	= ψ_{2L} coefficient for life load Q_L and pushover analysis
Static Gamma_QL	= γ_{QL} coefficient for life load and static analysis
Snow load	= snow load Q_s on the ceiling, typically should be applied for the top ceiling for buildings above 1000 m above sea level.
Pushover Phi_S	= φ_S coefficient for snow load Q_s and pushover analysis
Pushover Psi_2S	= ψ_{2S} coefficient for snow load Q_s and pushover analysis
Static Gamma_QS	= γ_{QS} coefficient for snow load and static analysis

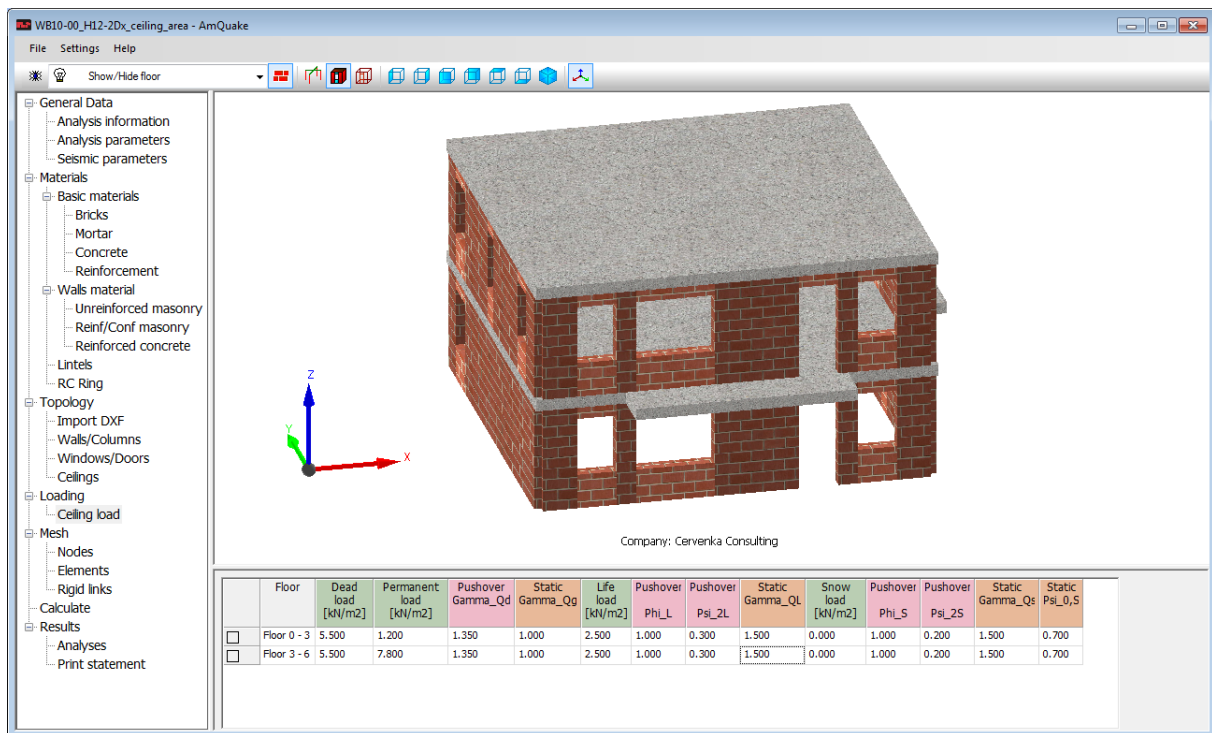


Figure 52: Ceiling load

3.6 Mesh

This item in the data input tree provides the possibility to define various parameters for the generation of the numerical model. The tables accessible from this data tree item provide information about the generated numerical model. It is possible to modify the generated finite element model. It is not recommended to make changes to the numerical model, since all changes will be lost when the model is regenerated as a response to various input data changes. Finite element model corresponding to the equivalent frame model is generated automatically by the application after clicking the **Generate mesh** button. Information about the mesh then appears in the next subsequent three items in the input data tree, i.e. nodes, elements, rigid links.

The mesh generation table contains the following information:

- Number of nodes** = number of nodes in the current model
- Number of elements** = number of elements in the current model
- Number of rigid links** = number of rigid links in the current model
- Min. overlapping O_min** = minimal overlapping of walls. This value is used by the program to automatically detect how the walls in upper floors are to be connected to the walls of the lower floors. When a top wall is overlapping the cross-section of a bottom wall by an area larger than $O_{min} \times t_1 \times t_2$, it is assumed that the two walls should be connected.

- Inter-storey connection** = this parameters influences how the connection of upper walls to the lower walls is evaluated. It is described in more detail in Section 5.2.2 of this manual. The following options are available:
- Direct:** corresponds to the direct rigid link from Sec. 5.2.3
- Proportional:** corresponds to the proportional rigid link from Sec. 5.2.3
- Semi direct:** it is a combination of the two above approaches.
- Rigid short beams** = when this parameter is selected the short vertical beams below parapets and above lintels are assumed to behave as rigid links. Additional rigid links are introduced into the model to simulate this effect. It should be noted that when this option is selected the internal forced in the short elements will be zero.
- Le/Lf** = relative coefficient of wall length, when the short beam will be rigid. If length of element $< (Le/Lf) * \text{wall Length}$, then the element will be rigid.
- Change rigid link to element** = Some models with the discontinuous walls in vertical direction are difficult to calculate. This option will replace rigid link with element with high E.
- Ceiling without bending stiffness** = by default this parameter is activated. This means that the diaphragm effect of the ceiling is considered only in the horizontal direction, i.e. in the plane of the ceiling, while the bending stiffness is assumed to be zero. When this parameter is not selected, full rigidity of the ceiling is assumed, i.e. including rigidity in bending.
- Change PM and RC to RM** = by default this parameter is non-activated. This parameter is useful for modelling confined masonry. If selected, program AmQuake automatically generates “RM” numerical model for confined masonry. The program assumes that the geometric model is built using the “CM” model. During the mesh generation the “CM” model is automatically converted into the “RM” model. For more information about “CM” and “RM” models for confined masonry see Section 4.2.
- XY smart reinforcement** = this checkbox appears only if the previous check box **Change PM and RC to RM** is activated. In the RM model the reinforcement of the confining element may be assigned to more than one wall. This may happen when the confining element is at the intersection of two walls. In this case, AmQuake normally assigns half of the reinforcement area to each wall using this confining element, but when the check box

XY smart reinforcement is activated the reinforcement is always assigned to the wall, which is significant for the given direction of the pushover analysis. This means that for the pushover analysis in X direction, the whole cross-sectional area of the reinforcement is assigned to the wall, which is parallel to the X direction, and zero reinforcement area is assigned to the walls that are parallel to the Y direction.

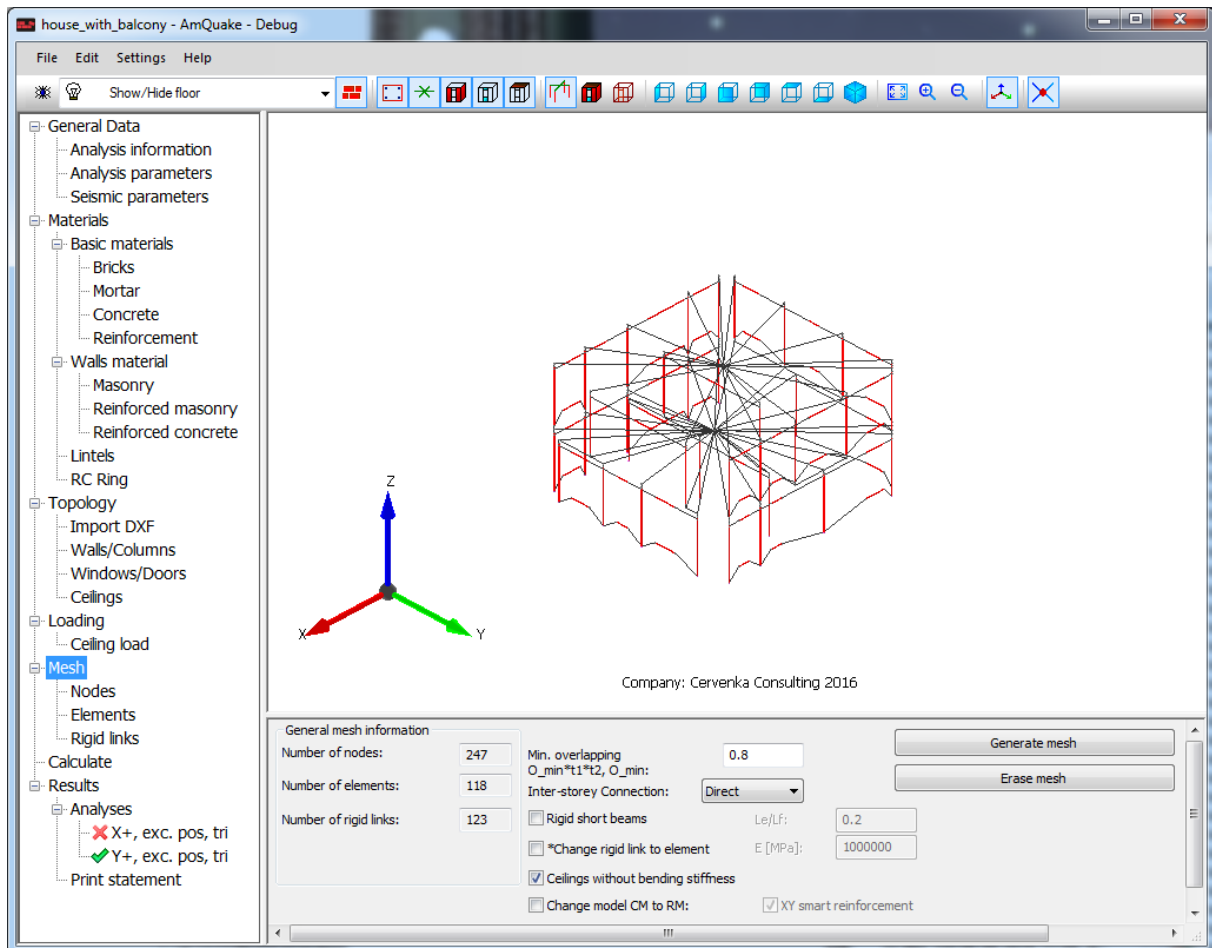


Figure 53: Generate Mesh

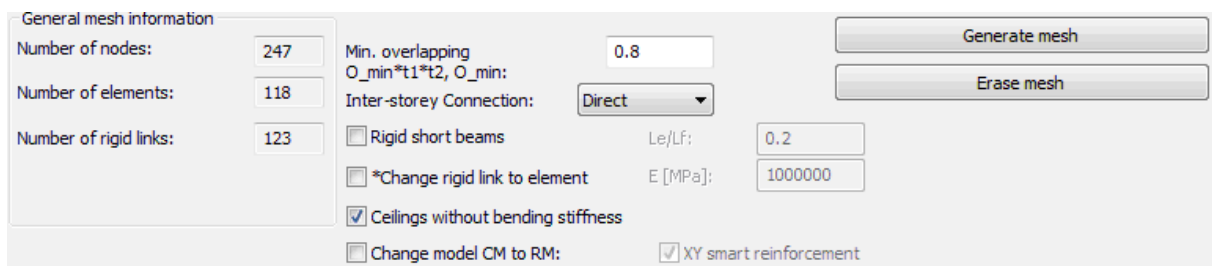


Figure 54: Generate Mesh Dialog when automatic Conversion of CM to RM model is selected

3.6.1 Nodes

This data item displays a table containing the list of finite element nodes in the numerical model.

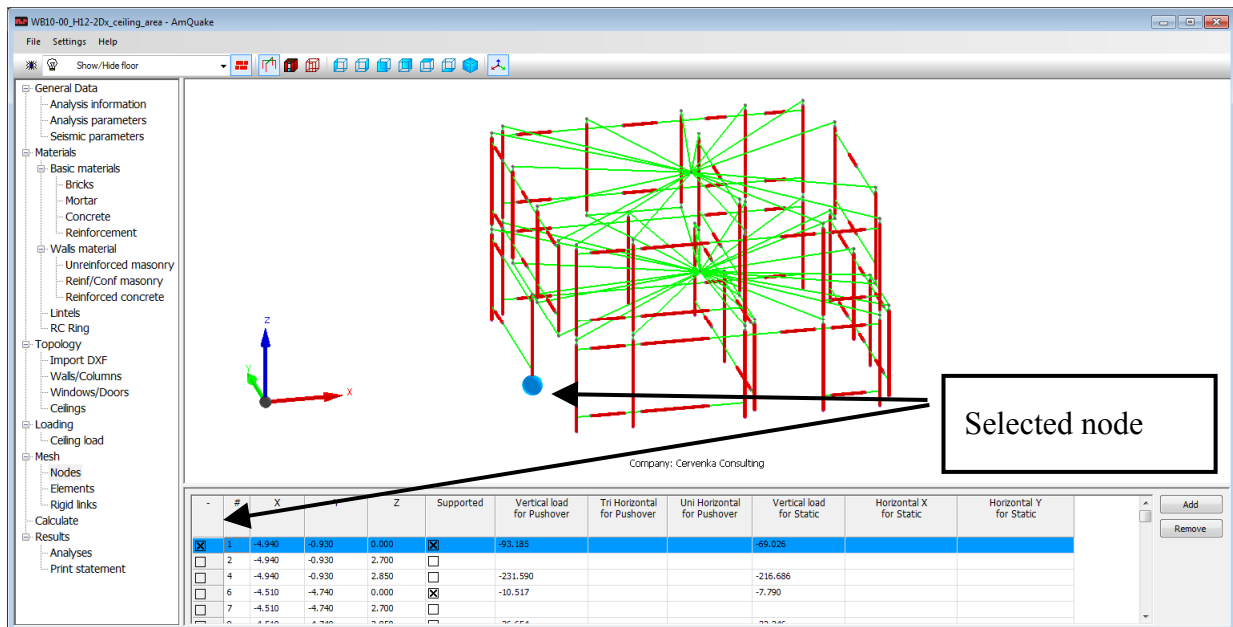


Figure 55: Mesh - Nodes

Number	= node number
X	= x-coordinate
Y	= y-coordinate
Z	= z-coordinate
Supported	= checkbox indicating that the given node is supported. Supported nodes are fixed in all directions and in all degrees of freedom.
Vertical load for pushover	= vertical force for pushover analysis
Tri Horizontal for pushover	= triangular horizontal force for pushover analysis
Uni Horizontal for pushover	= uniform horizontal force for pushover analysis
Horizontal X for static	= horizontal forces for static analysis
Horizontal Y for static	= horizontal forces for static analysis

3.6.2 Elements

Information about the elements of the generated mesh

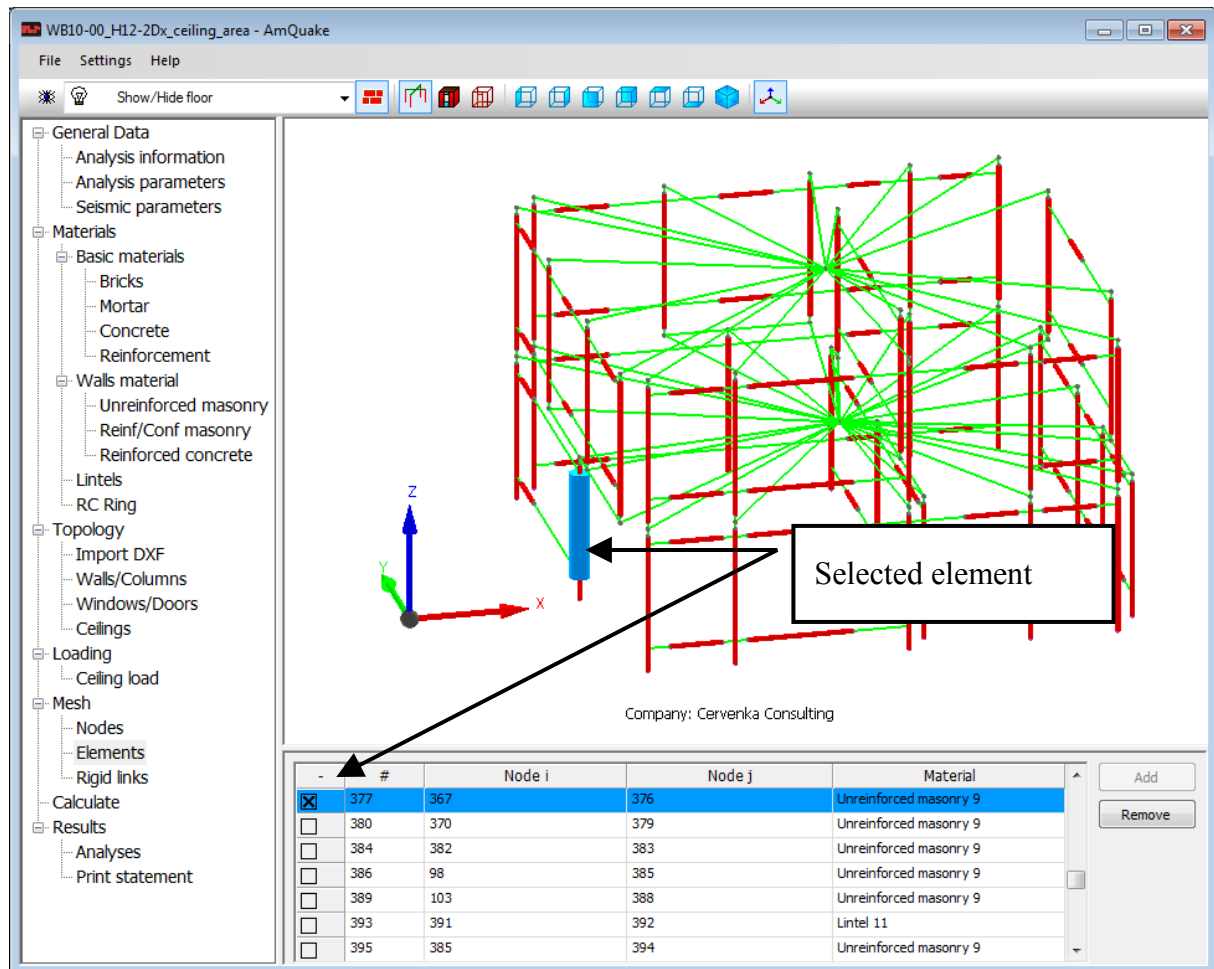


Figure 56: Mesh - Elements

Number = element number
Node i = first element node
Node j = second element node
Material = element material

3.6.3 Rigid links

This data tree section contains the information about the rigid links in the numerical model. The theory behind these links is described in Section 5.2.2.

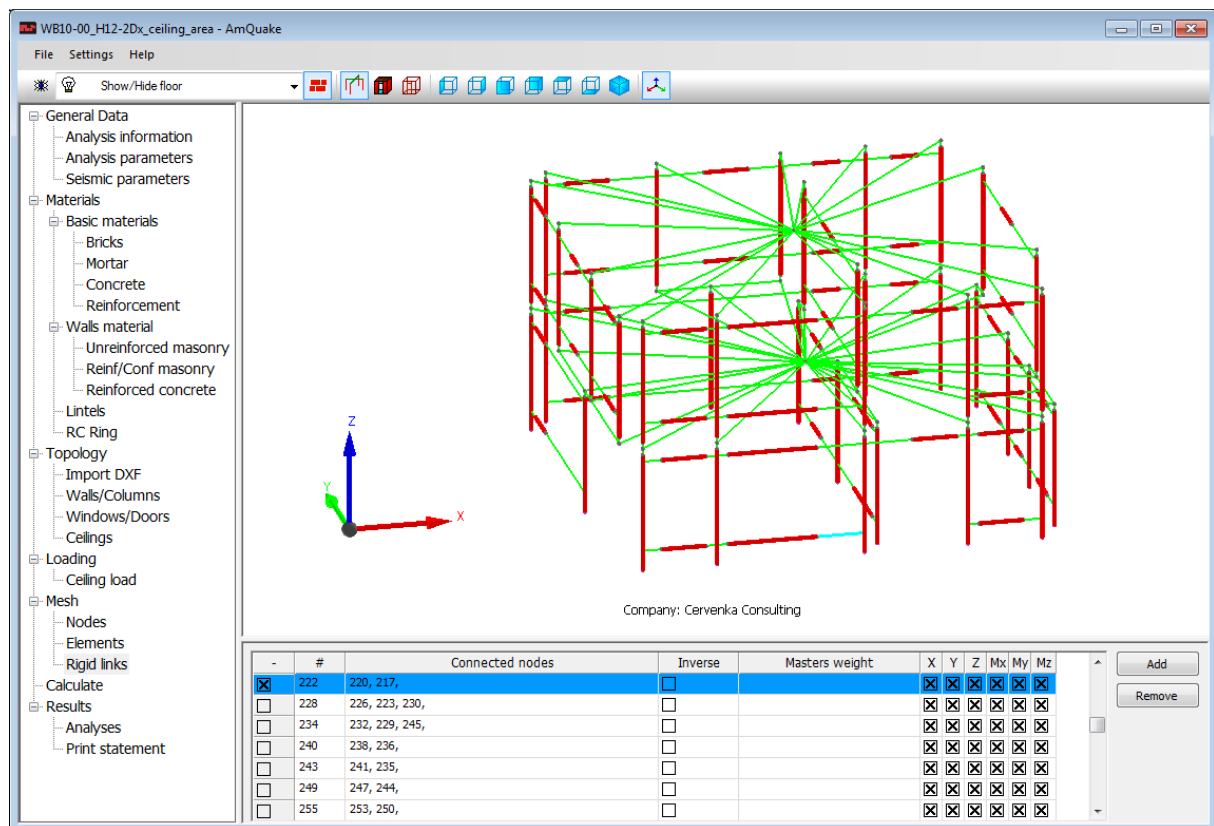


Figure 57: Mesh – Rigid lines

Number = rigid link id

Inverse = checkbox identifying an inverse rigid link (see Section 5.2.2)

Masters weights = weight coefficients of the individual master nodes. This option is used on in connection with the inverse checkbox.

Connected degrees of freedom = these checkboxes identifies, which degrees of freedom are activated by this rigid link.

3.7 Calculate

This item in the input data tree opens a dialog window at the bottom of the program screen. Here a user can modify various parameters for the pushover analysis as well as it is possible to control, which analyses will be performed.

The first property sheet **Analyses** contains a list of possible analyses. Individual analyses can be activated by selecting an appropriate check box. Each column in the table represents a pushover analysis in one direction. EC8 requires that pushover analysis is performed at least in two perpendicular directions. It also requires that two types of horizontal force distribution are considered: triangular and uniform. This means that in order to satisfy EC8 requirements all checkboxes need to be selected. The program **AmQuake** can be used also to perform a static assessment of the vertical walls. The static analysis is activated by selecting the **Static** check box in the right part of this dialog.

The second property sheet contains various **Advanced settings** for the pushover analyses.

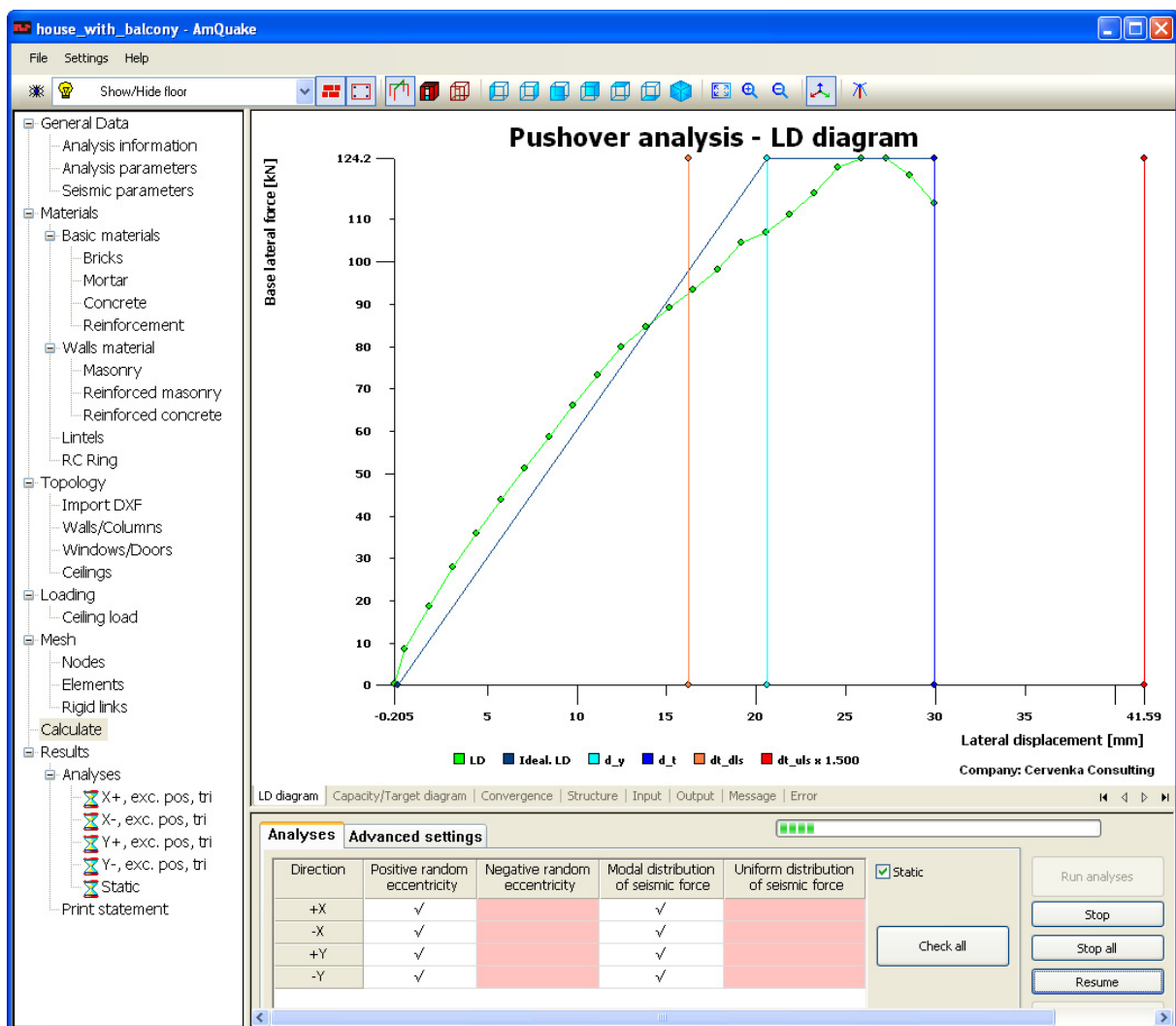


Figure 58: Calculate – Analyses

Figure 59: Calculate – Advanced Settings

Solver values:

Energy tol. = energy tolerance

Equilibrium tol. = force tolerance, i.e. relative error in the force equilibrium

Displacement tol. = displacement tolerance, i.e. relative value of maximal iterative displacement correction

Max. force tol. = seismic force tolerance. This value represents a tolerance for the iterative process that is expressed as a relative value with respect to the total value of the applied forces.

Step settings:

Max. number of steps = maximal number of steps in each analysis

Max. number of iterations = maximal number of iterations in each step

Required no. of iterations = optimal number of iterations, steps size is increased or reduced based on this parameter.

Analysis:

Pause analyses = analysis will be paused after each step and task in the calculation process. For instance the program will also stop after generating the input for the program finite element core. This input file can be investigated under the input data tree item **Calculate | Input**.

Number of steps for vertical load = number of steps to apply the initial vertical load. This also affects the step size in the subsequent pushover analysis. If larger value is specified the vertical load will be applied by smaller steps. Subsequently, also smaller steps will be used in the pushover analysis.

Min. rel. step Length = this value limits the reduction of the step size during the pushover analysis. During the pushover analysis the step size may be reduce in order to satisfy the specified optimal number of iterations

	(see the parameter Required no. of iterations). In some cases the step size may be reduced to such low values that the analysis will be very slow. The default value -1 indicates that no limit is specified. When the analysis becomes too slow, and program is using very small time steps, it is recommended to set this value to 1.
Max. rel. step length	= this value limits the increase of the step size during the pushover analysis. During the pushover analysis the step size may be increased in order to satisfy the specified optimal number of iterations (see the parameter Required no. of iterations). In some cases the step size may be increased so much that the whole pushover analysis may be performed in just few steps. The default value -1 indicates that no limit is specified. When convergence problems occur during the analysis, and large errors are reported, it is recommended to adjust this value. Typical values could be for instance in the range between 1 to 10. The convergence should be improved, when this parameter is set to a value closer to 1.
Step length	= this parameter activates the manual definition of the step size for the push over analysis. By default the step length is prescribed by defining the displacement of the top floor. The default step length is calculated such that the program will need approximately 50 steps before reaching a maximal value of one of the drift limits, as they are defined in Analysis parameters Advanced settings (see Section 3.3.2). This parameter should be activated in case of convergence problems and the value of the step length should be reduced to a lower value.
Only top master displ.	= this parameter identifies how many nodes (i.e. points) of the numerical models should be used for the calculation of the step length. In the default settings, only the displacement of the top floor is used. In certain situations, this may result in convergence problems. When this happens, this check box may be deselected. Then the displacements of the whole model will be used in the arc-length solution method.
P_f to stop analysis	= this input field defines the relative value of the peak base shear force, which will cause the pushover analysis to terminate. When the base shear force drops below this value relative to the peak shear force, the analysis will stop, and the

program will continue with pushover analyses in other directions.

Stop after

= this input field causes the analysis to terminate as soon as the program recognizes that the ULS criterion is satisfied. **ULS and DLS** option the program may continue with the nonlinear analysis even after satisfying ULS and DLS criterion. **ULS** option will analysis even to satisfying only ULS criterion. **Failure** option will analysis to total failure of construction (Base Lateral force < P_f)

Use Best Iteration

= this input field will use iteration with lowest error. This feature improves the accuracy and stability of the nonlinear analysis.

Use Step Reduction Allowance

= when this field is selected, the steps that have not converged will be repeated with reduced step size. This feature improves the accuracy and stability of the nonlinear analysis.

Default values button

= this button will reset all the parameters in this dialog to their default values

3.8 Results

This section displays all the results from the completed analyses. User directly sees in the left tree, which analyses have been verified. The analyses that satisfied the EC8 code (or P100 in case of Romania) requirements are denoted by “✓” sign. The analyses that failed are indicated by the red cross sign ✗.

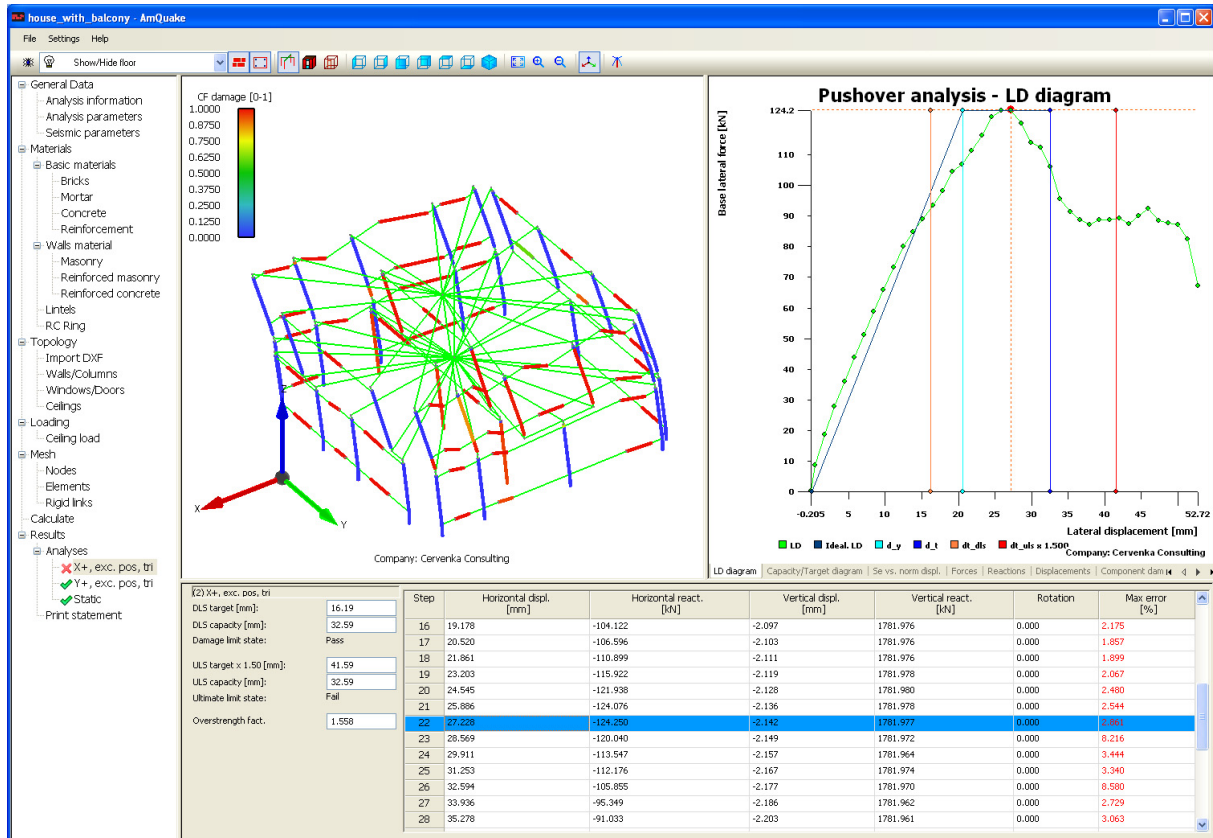


Figure 60: Results – Analyses

3.9 Analysis

The item **Analysis** can be used to access the results from the individual analyses. A user can explore specific analysis details or examine the summary from all analyses. This summarizing table is displayed by selecting the item **Results | Analyses**, which shows the table as described in Figure 61. This table summarizes the main results and parameters from the individual pushover analyses. The meaning of the individual parameters is the following:

Limit state

= information whether DLS and ULS requirement was satisfied or not. The following answers are possible:

Pass – DLS and ULS requirements are satisfied, it means that DLS target is smaller than DLS capacity and ULS target is smaller than ULS capacity.

Fail – DLS and ULS requirement is not satisfied, it means that at least one of the two criteria (i.e. ULS or DLS are not satisfied).

Unknown – the pushover analysis was interrupted before finding a solution. This could be due to user intervention or other error during the pushover analysis.

DLS target

= top floor displacement, at which the inter-storey drifts for each floor are checked if they do not violate the damage limitation requirement. The inter-storey drift for each floor should be smaller than the **Damage limit** value provided in Figure 18. For more information see also the theoretical description in Section 5.1

DLS capacity

= top floor displacement when the inter-storey drift for a certain floor violates the damage limitation requirement. The inter-storey drift for each floor should be smaller than the **Damage limit** value provided in Figure 18. For more information see also the theoretical description in Section 5.1

ULS target

= top floor displacement that can be expected for the given structure and seismic demand. The structure needs to be able to sustain such top floor displacements without collapse. For more information see also the theoretical description in Section 5.1

Name	Limit states	DLS target [mm]	DLS capacity [mm]	ULS target x 1.50 [mm]:	ULS capacity [mm]	Safety margin DLS [%]	Safety margin ULS [%]	Max error [%]	Storey
X+, exc. pos, uni	Pass	0.047	0.165	0.121	0.165	71.404	26.551	0.442	1
X+, exc. pos, tri	Pass	0.047	0.165	0.121	0.165	71.404	26.551	0.442	1
X+, exc. neg,	Pass	0.047	0.165	0.121	0.165	71.404	26.551	0.442	1
X+, exc.neg, tri	Pass	0.047	0.165	0.121	0.165	71.404	26.551	0.442	1
X-, exc. pos, uni	Pass	0.047	0.165	0.121	0.165	71.404	26.551	0.442	1
X-, exc. pos, tri	Pass	0.047	0.165	0.121	0.165	71.404	26.551	0.442	1

RShift	Dls Criterion	RShift Criterion	Uls Criterion	Dls Step Id	Uls Step Id	Period T Norm.	Max reference ground acceleration	Ductility	Overstrength fact.
0.000	3.497	0.015	1.361	4	4	0.049	1.225	1.246	1.000
0.000	3.497	0.015	1.361	4	4	0.049	1.225	1.246	1.000
0.000	3.497	0.015	1.361	4	4	0.049	1.225	1.246	1.000
0.000	3.497	0.015	1.361	4	4	0.049	1.225	1.246	1.000
0.000	3.497	0.015	1.361	4	4	0.049	1.225	1.246	1.000
0.000	3.497	0.015	1.361	4	4	0.049	1.225	1.246	1.000

Figure 61: Results – Analyses – Table with results

ULS capacity

= top floor displacement when the structure starts to collapse or when the base shear force drops below 80% of its maximal value. AmQuake program does not always performs the pushover analysis up to full collapse. The analysis is interrupted, when the top floor displacements are larger than the required target. For more information see also the theoretical description in Section 5.1

Safety margin DLS	= defines the safety margin for DLS in % of the DLS capacity, i.e. $100 * (DLS\ capacity - DLS\ target) / DLS\ capacity$.
Safety margin ULS	= defines the safety margin for ULS in % of the ULS capacity, i.e. $100 * (ULS\ capacity - ULS\ target) / ULS\ capacity$.
Max error	= maximal convergence error from all step from up to the step when the ULS target is reached.
Storey	= the storey id where the inter-storey drift limit was reached.
RShift	= calculated maximum inter-storey drift from all storey's deflection / storey height (within the step of storeys and executed steps)
DLS Criterion	= DLS pushover criterion ratio DLS capacity/DLS target
Rshift Criterion	= DLS inter-storey drift criterion Rshift/Damage limit
ULS Criterion	= ULS pushover criterion ratio ULS capacity/ULS target
DLS Step Id	= Step id when the inter-storey damage limit was reached. The storey id is indicated in the parameter Storey .
ULS Step Id	= Step id when the ULS capacity was reached
Period T Norm.	= Period of vibration of the idealized structural system with single degree of freedom
Max reference ground acceleration	= Maximal possible reference ground acceleration for the analysed model. For the given structural geometry, loading direction, selected spectrum type provides the value of the maximal reference ground acceleration the structure will be able to survive.
Ductility	= target displacement / elastic displacement
Overstrength factor	= this value can be used to determine the q factor for the analysed structure. It is the ratio of the peak value of the shear force with respect to the base shear force value when the first plastic hinge developed in one of the vertical elements.

The main window (see Figure 60) for each analysis result is split into two panels. The left panel is used for rendering the model of the construction with capabilities of displaying the deformations, forces and damages. The right panel contains various graphs as well as hard numerical data for each node or finite element. The two windows contain results from the analysis that is selected in the data input tree on the left side. Each analysis consists of many load steps. The load steps are summarized in the table at the bottom part of the program screen. The results for an individual step are selected by highlighting an appropriate row in the list of analysis steps. When a certain step is selected, the content of the two top windows changes, and they display the graphical or numerical results for the selected step.

The window, which is located next to the load step tables (see Figure 62) provides the summary of the pushover curve evaluation for the selected analysis. DLS and ULS limits states are evaluated here as well as the overstrength factor.

(1) X+, exc. pos, uni	
DLS target [mm]:	0.0471
DLS capacity [mm]:	0.165
Damage limit state:	Pass
ULS target x 1.50 [mm]:	0.121
ULS capacity [mm]:	0.165
Ultimate limit state:	Pass

Figure 62: Summary of the Pushover Analysis Evaluation

Damage limit state = information whether DLS requirement was satisfied or not. The following answers are possible:

Pass – DLS requirement is satisfied, it means that DLS target is smaller than DLS capacity.

Fail – DLS requirement is not satisfied, it means that DLS target is larger than DLS capacity.

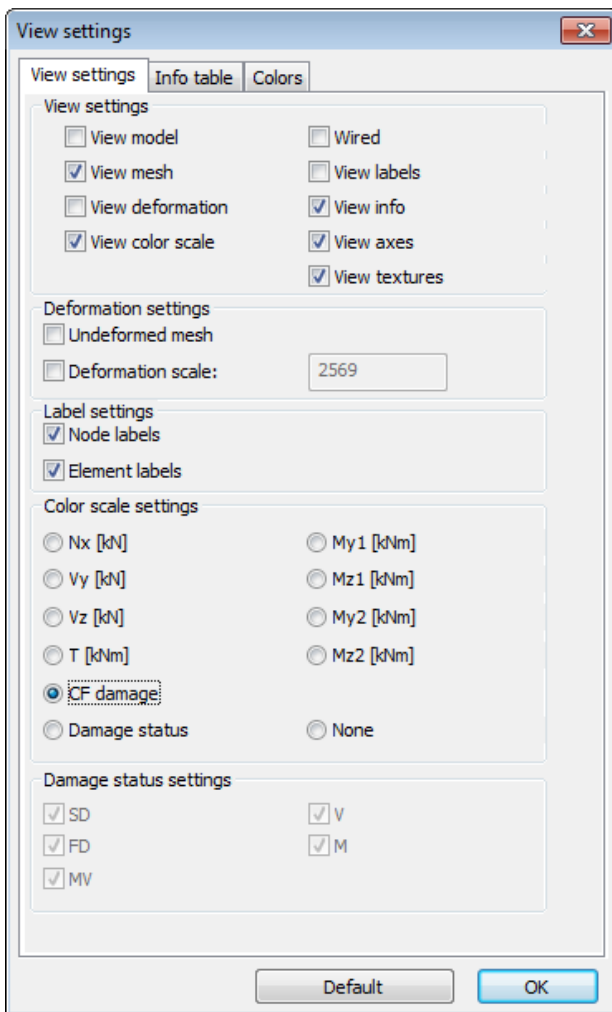
Unknown – pushover analysis was interrupted before finding a solution.

Ultimate limit state = information whether ULS requirement was satisfied or not. The following answers are possible:

Pass – ULS requirement is satisfied, it means that ULS target is smaller than ULS capacity.

Fail – ULS requirement is not satisfied, it means that ULS target is larger than ULS capacity.

Unknown – pushover analysis was interrupted before finding a solution.




CF_damage = parameter indicating how much the element response deviate from the elastic one.

Damage status = shows how the element forces relate to the ultimate capacities:

- SD** - total shear capacity exhausted, full shear failure
- FD** - total flexural capacity exhausted, full flexural failure
- MV** - both moment and shear ultimate forces are reached, plastic behaviour in flexure and shear
- M** - moment capacity reached, plastic behaviour in bending/flexure
- V** - shear capacity reached, plastic behaviour in shear

Figure 63: Dialog for the Setting of various Visualization Options. This dialog is opened using the button 

There are numerous features for the visualisation of results from the calculated analysis. The results can be visualized graphically in the middle window with the structural model. The contents of this window can be modified through the View settings using the button  or by the View settings item in the contextual menu, which is activated by the right mouse button in the graphical window.

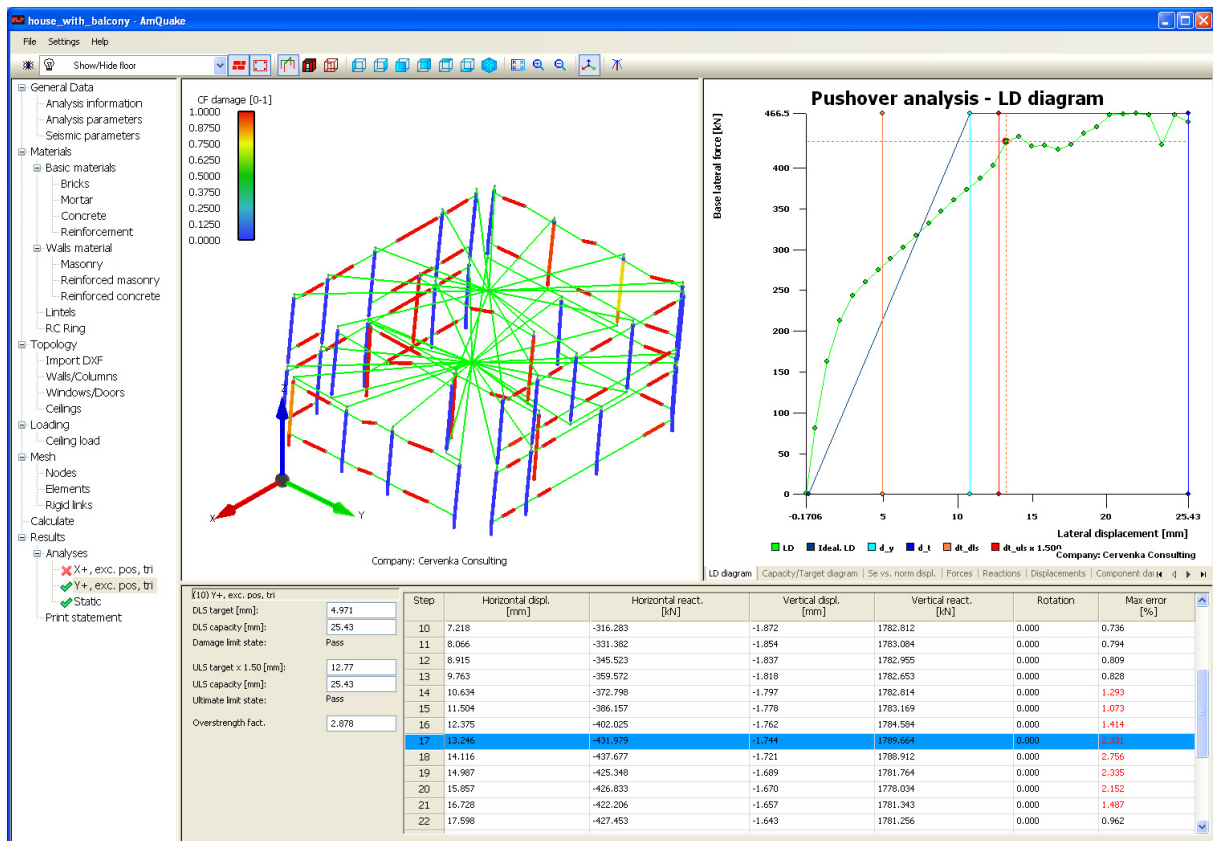


Figure 64: Results – Analyses – Analysis 2

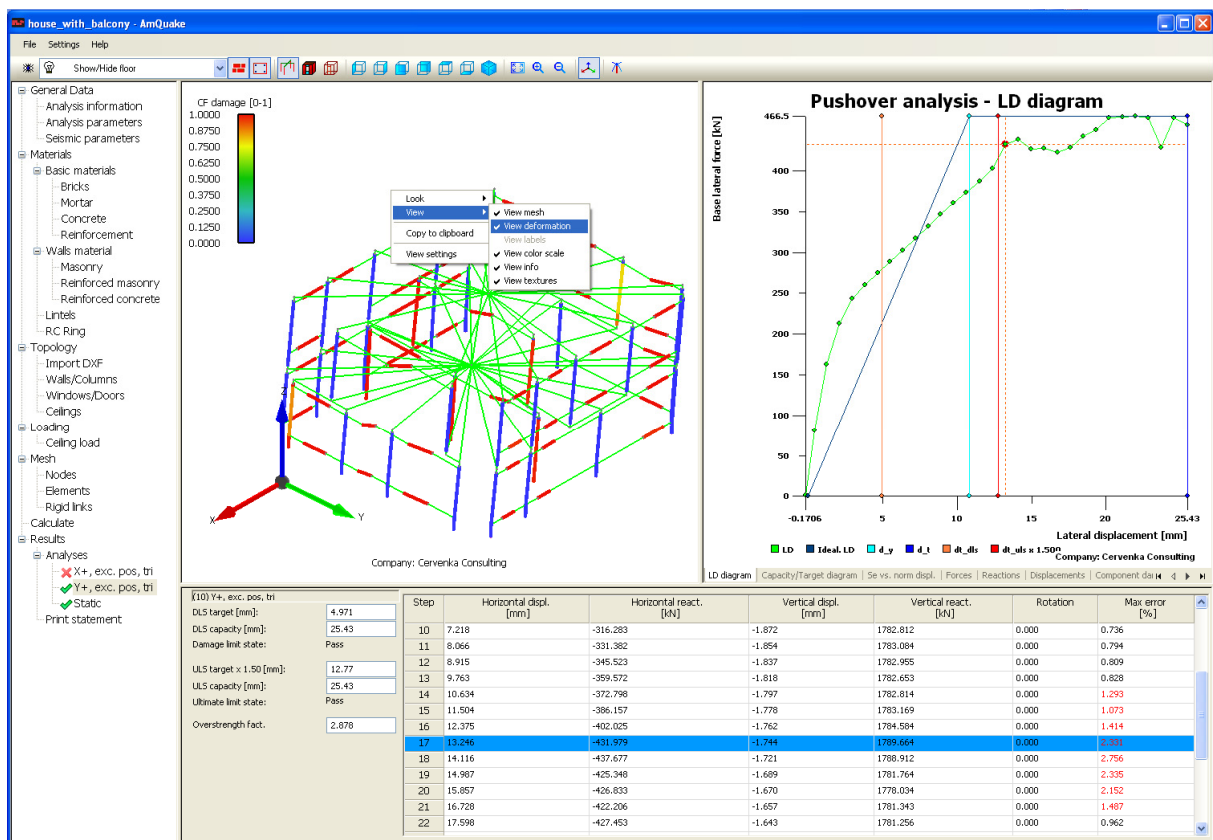


Figure 65: Selection of Deformed Shape and Visualization of Damaged Elements

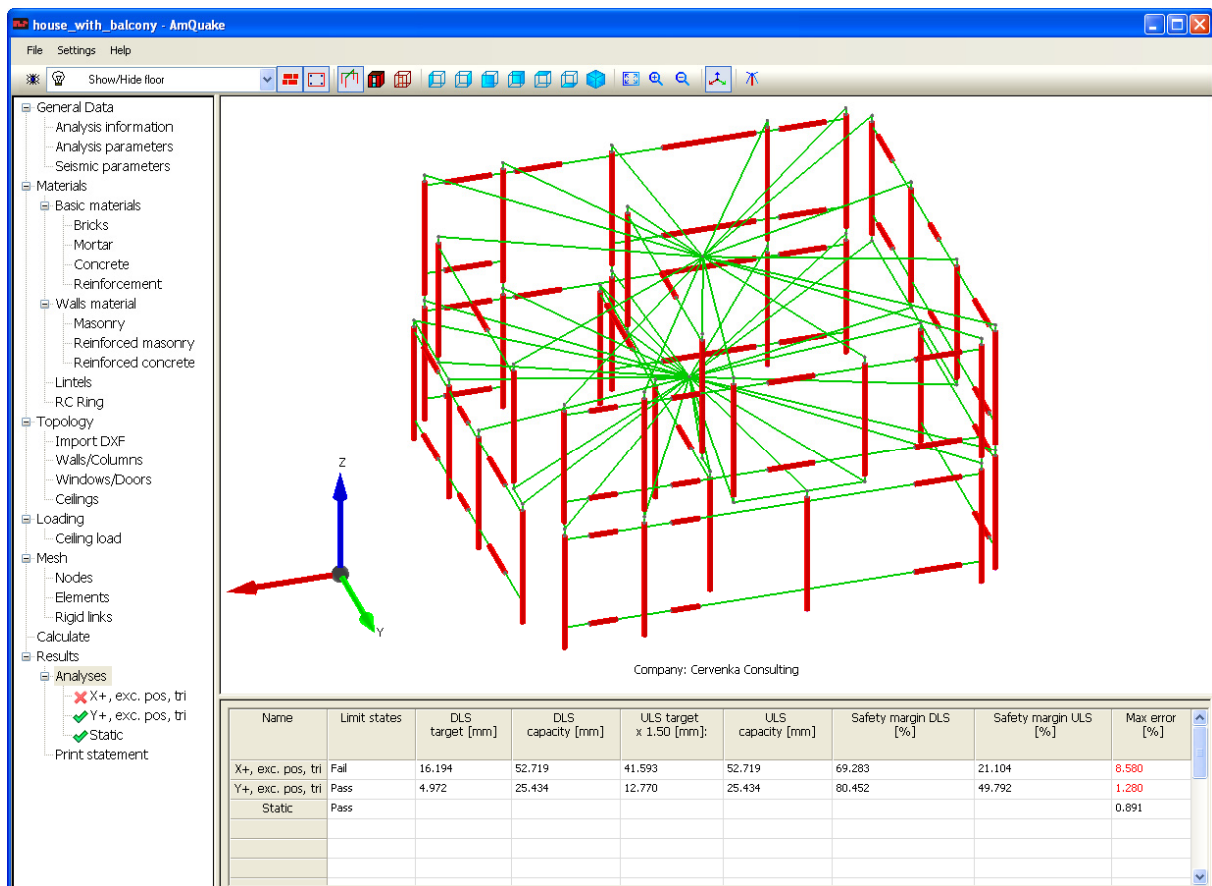


Figure 66: Summary of Analysis Results

The summary of the results from all analyses can be obtained by selecting the item Analyses. This shows a table where the main results from each analysis are listed. The last column in this table gives the information about the possible solution error. This error indicates whether the nonlinear solver was able to find a solution, and the accuracy of the calculated results. When the error is above 10%, the calculation parameters should be adjusted to improve the results.

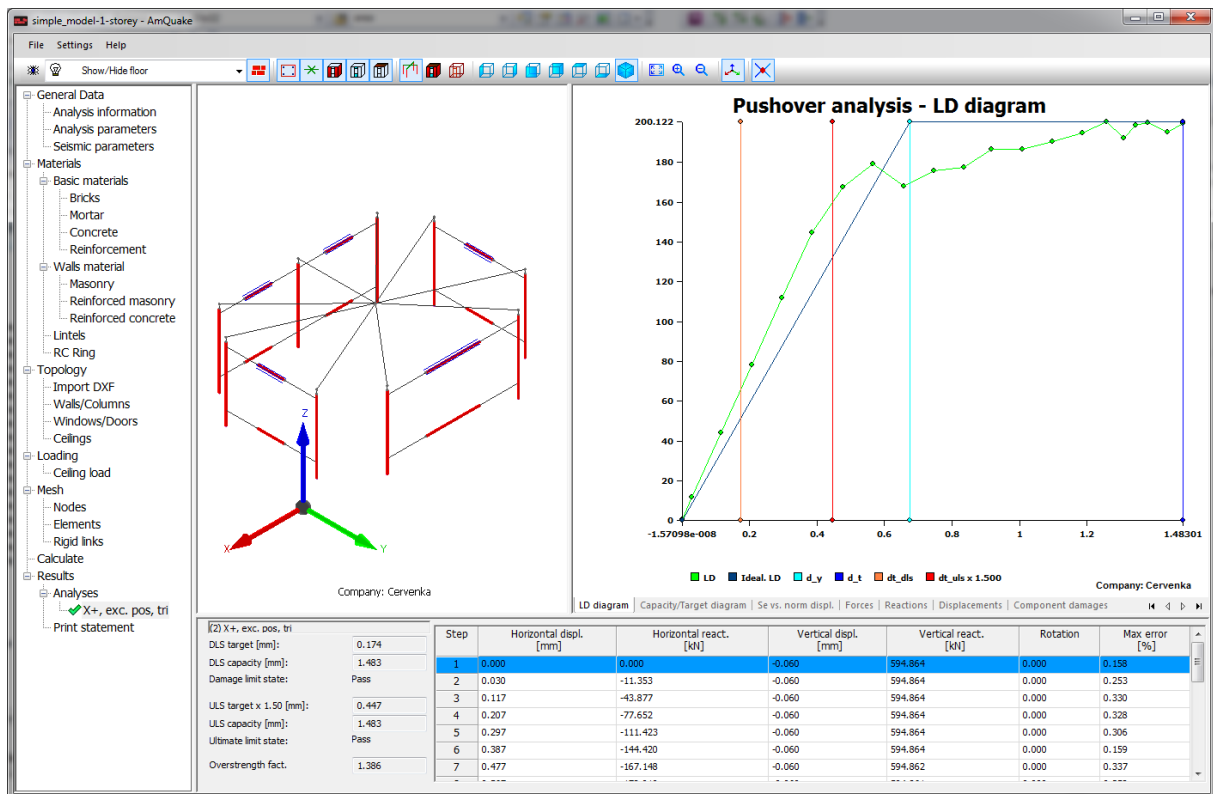


Figure 67: Results: LD diagram

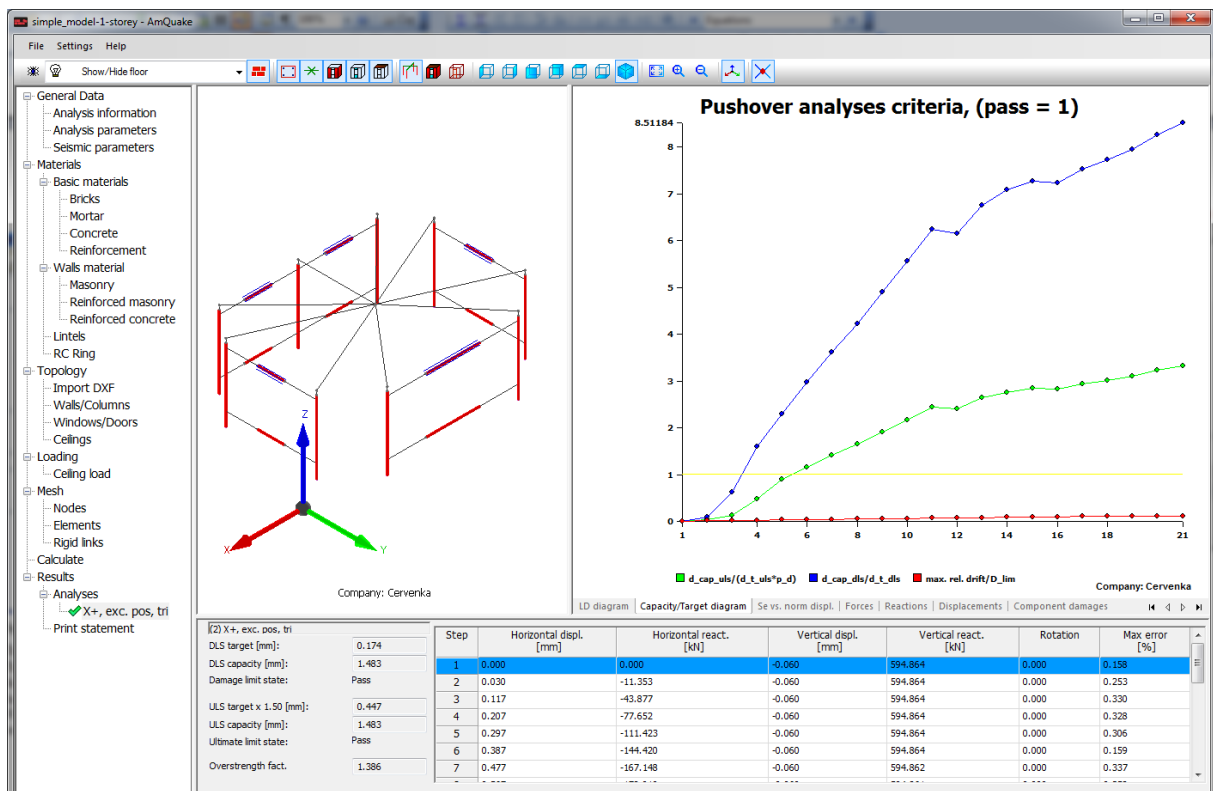


Figure 68: Results: Capacity / Target diagram

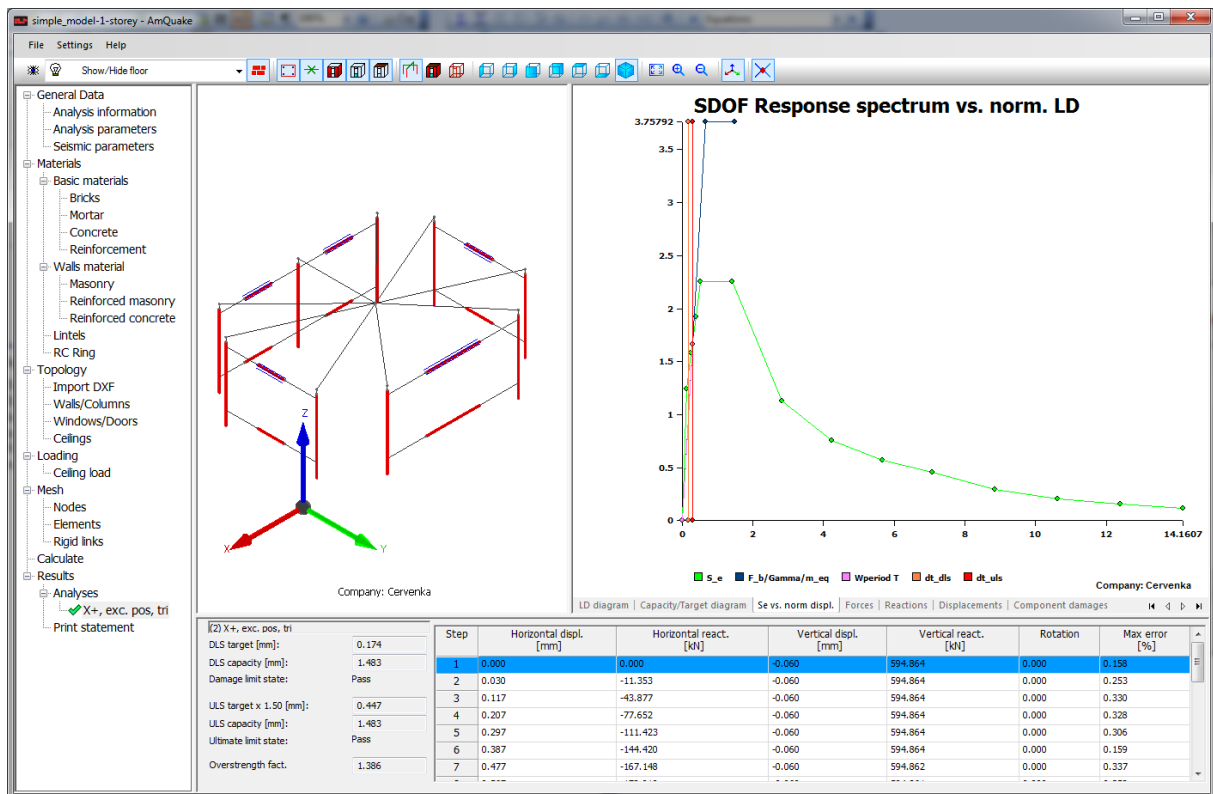


Figure 69: Results: Se vs. norm displacements

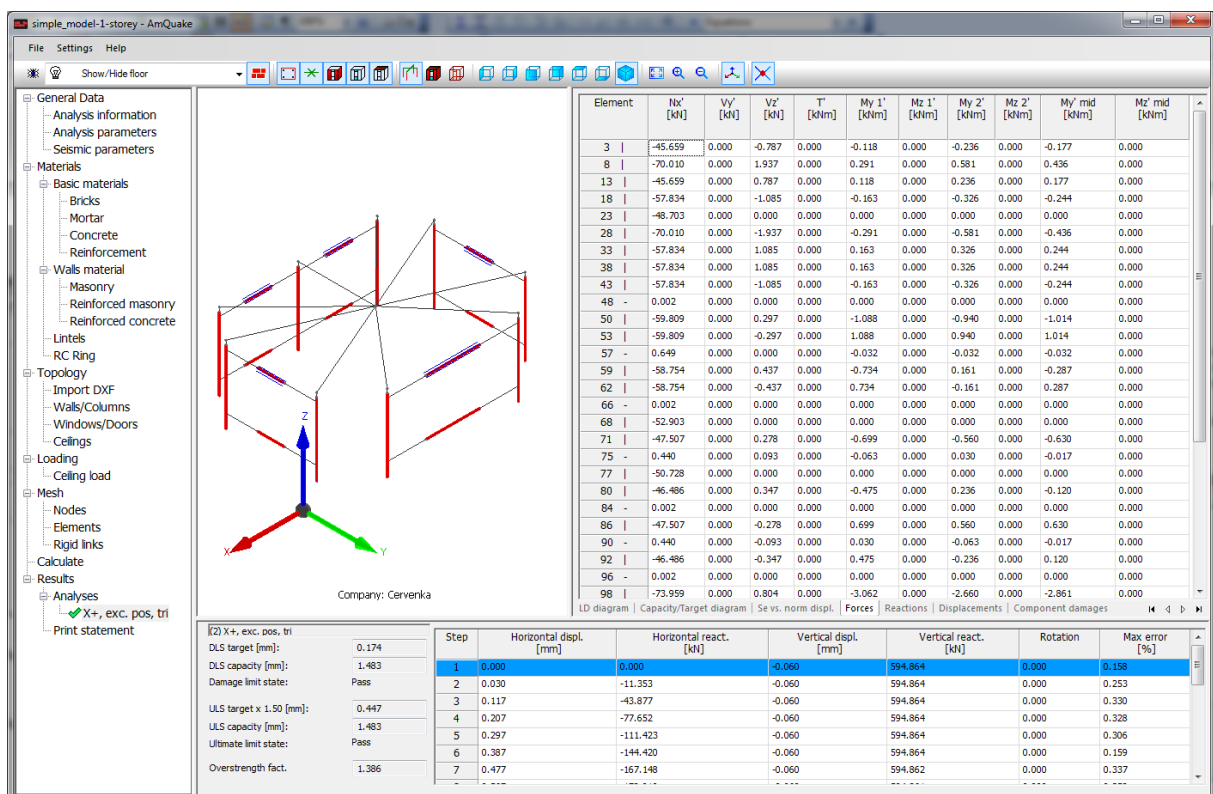


Figure 70: Results: Forces

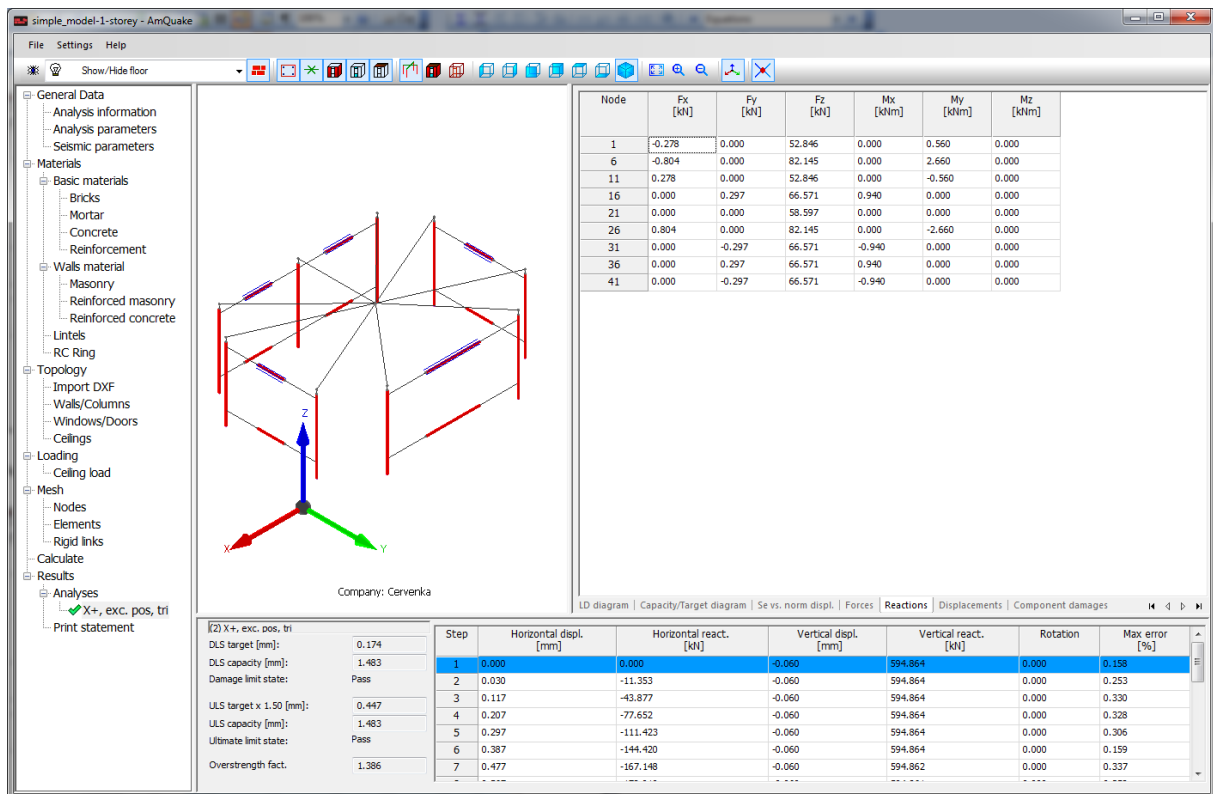


Figure 71: Results: Reactions

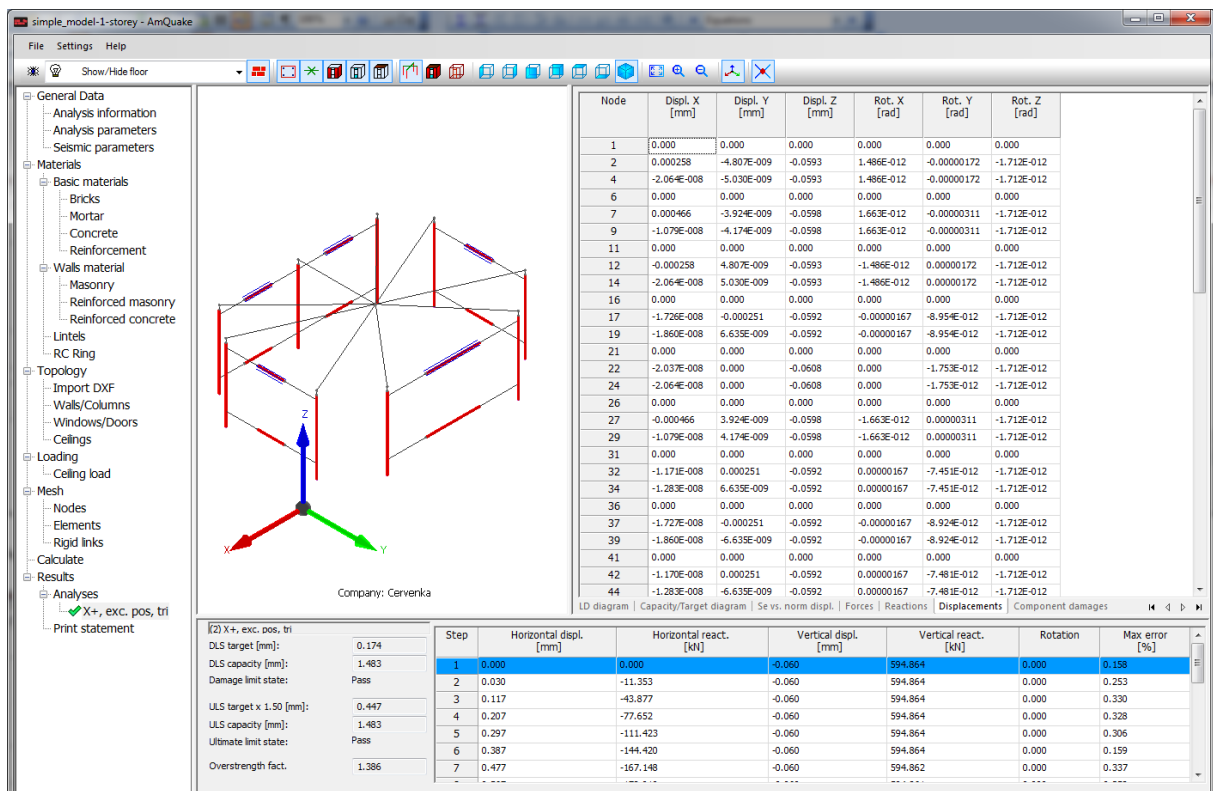


Figure 72: Results: Displacements

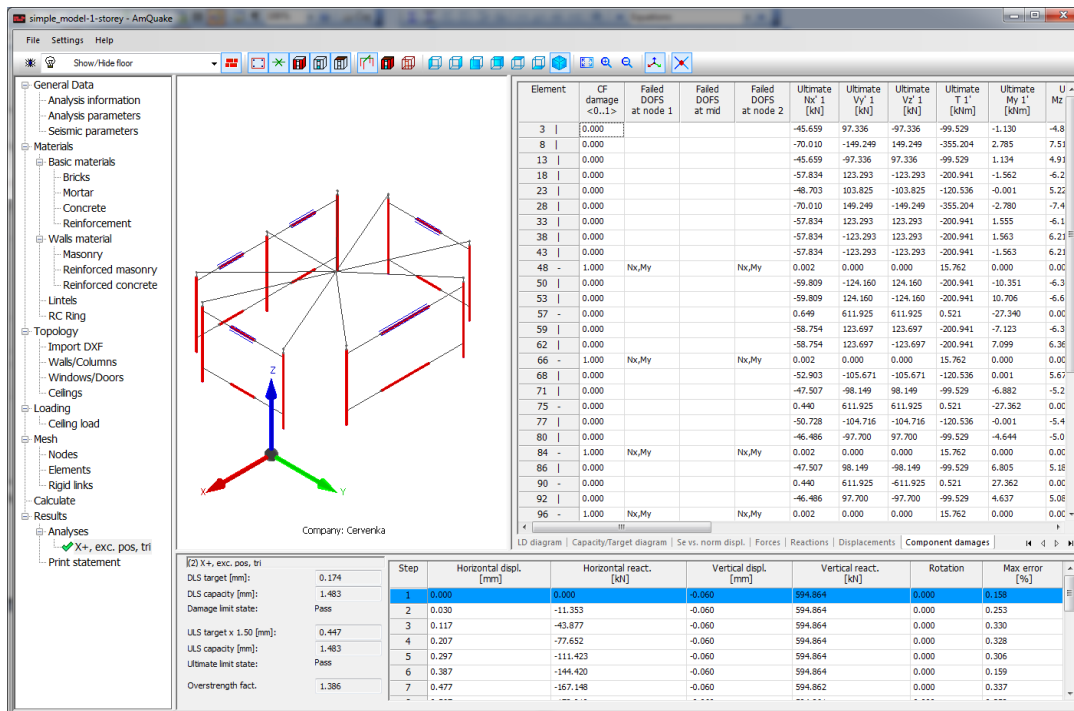


Figure 73: Results: Components damages

The component damages indicate the status of individual elements:

CF damage: this parameter indicates how much the beam behaviour deviates from the elastic response

Failed DOFs: indicates, which cross-sectional force has reached the ultimate moment. Label SD indicates full shear failure, i.e. full deformation capacity in shear is exhausted. The beam cannot carry any shear force. Label FD indicates full flexural failure, i.e. full deformation capacity in bending is exhausted, and the beam cannot carry any moment.

Ultimate forces: Ultimate N_x' is N_{rd} according the interaction diagram in Figure 74. Ultimate M_y' and Ultimate M_z' are projection of $M_{y'rd}$ from Figure 74. The construction of this interaction diagram is described in Section 5.3.

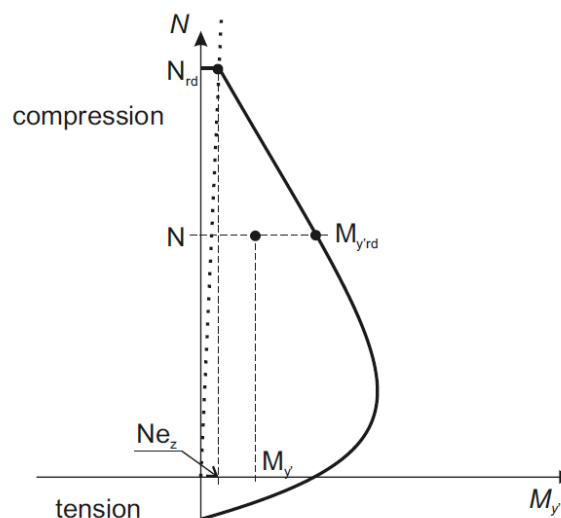


Figure 74: Results: M-N diagram

3.10 Print statement

In this dialog, user can set the properties of printed statements. User can choose between two predefined settings (brief protocol and complete protocol) or set the properties manually. Printing complete protocol contains all information about the project including materials, topology, loadings as well as all the analyses results.

The created protocol can be saved into an rtf file, which can be further edited by various word processors.

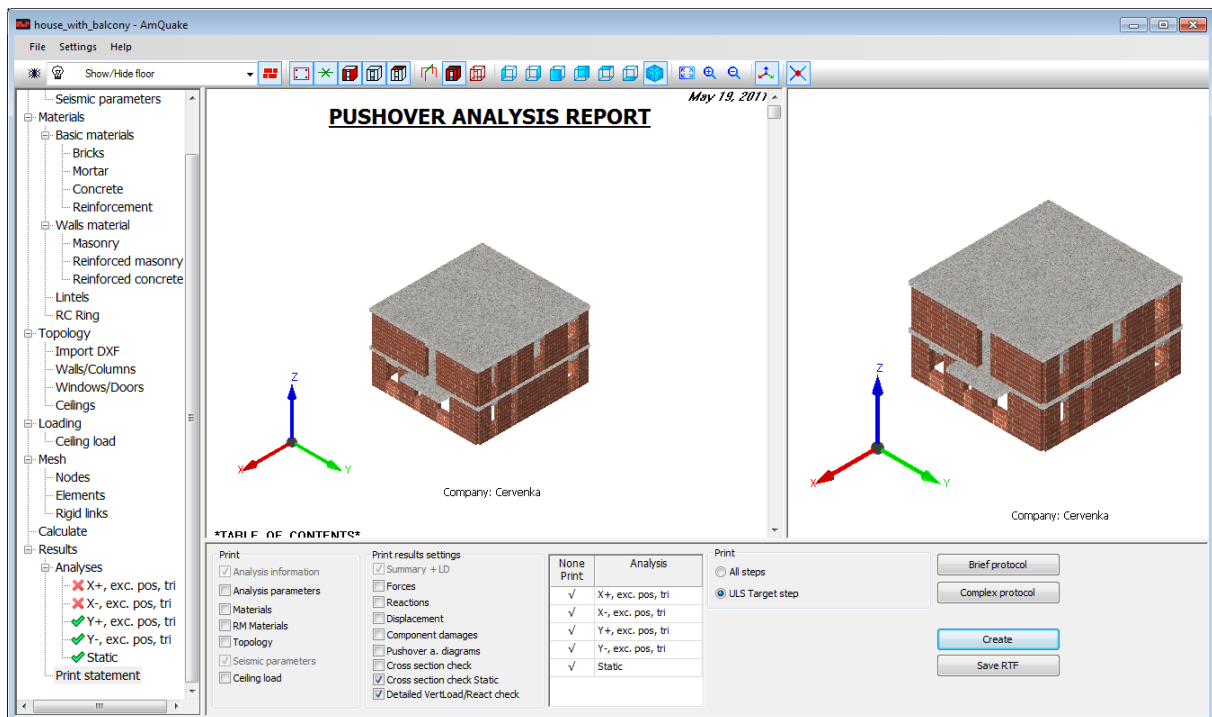


Figure 75: Print statement

4 HOW TO DO

This chapter provides guideline how to model various specialized elements of masonry buildings in the program **AmQuake**.

4.1 How to model columns

The columns are modelled as walls with rectangular or square cross-section.

4.2 How to model confined masonry

The confined masonry consists of masonry walls that are confined by reinforced concrete vertical elements. These reinforced concrete elements should be along each opening and they should be included in each wall longer then certain length. The national annex should be consulted for the exact requirements for the reinforced concrete elements. In **AmQuake** there are several possibilities how to model confined masonry:

- (a) Single wall with reinforced masonry material and vertical reinforcement located at the required locations (Model RM)
- (b) Divide each wall into several walls/columns, edge walls are using material reinforced masonry while the walls between them are from unreinforced masonry (Model RM2).
- (c) Divide each wall into several walls/columns, edge walls are defined with reinforced concrete material while the walls between them are defined with unreinforced masonry (Model CM).

It is recommended to use the DXF import capabilities. Prior to defining the walls in the DXF file, a decision should be made, which of the above approaches shall be used. The walls and columns should be defined in the DXF file with respect to the selected method of modelling the confined masonry. This means that if method (b, i.e. RM2) or (c, i.e. CM) is to be used, it is necessary to divide each wall into several walls and columns as it is shown in Figure 79 to Figure 81. The columns will be later assigned with the reinforced masonry or reinforced concrete materials, while the walls between them will be assigned an appropriate masonry material.

Generally it can be said that model RM, i.e. model (a) will provide higher seismic capacity as described in the report [12]. RM model is also more in the line with the current engineering practice for the calculation of confined masonry shear as well as bending strength. The definition of this model in **AmQuake** is slightly more demanding, and there is a problem that the confining elements may be duplicated in the locations where there is a connection of several walls. In this case, it is necessary to make an appropriate reduction of the cross-sectional area of the reinforcement in the overlapping regions.

Models RM2 and CM are easier to define in AmQuake, and there is no problem with the overlapping confining elements. The disadvantage of this approach is that it underestimates the shear strength of masonry, and thus may result in very conservative conclusions regarding the seismic safety.

AmQuake enables automatic conversion from CM model to RM model (see Figure 78 and Section 3.6), this greatly simplifies the definition of the RM model in practical analyses.

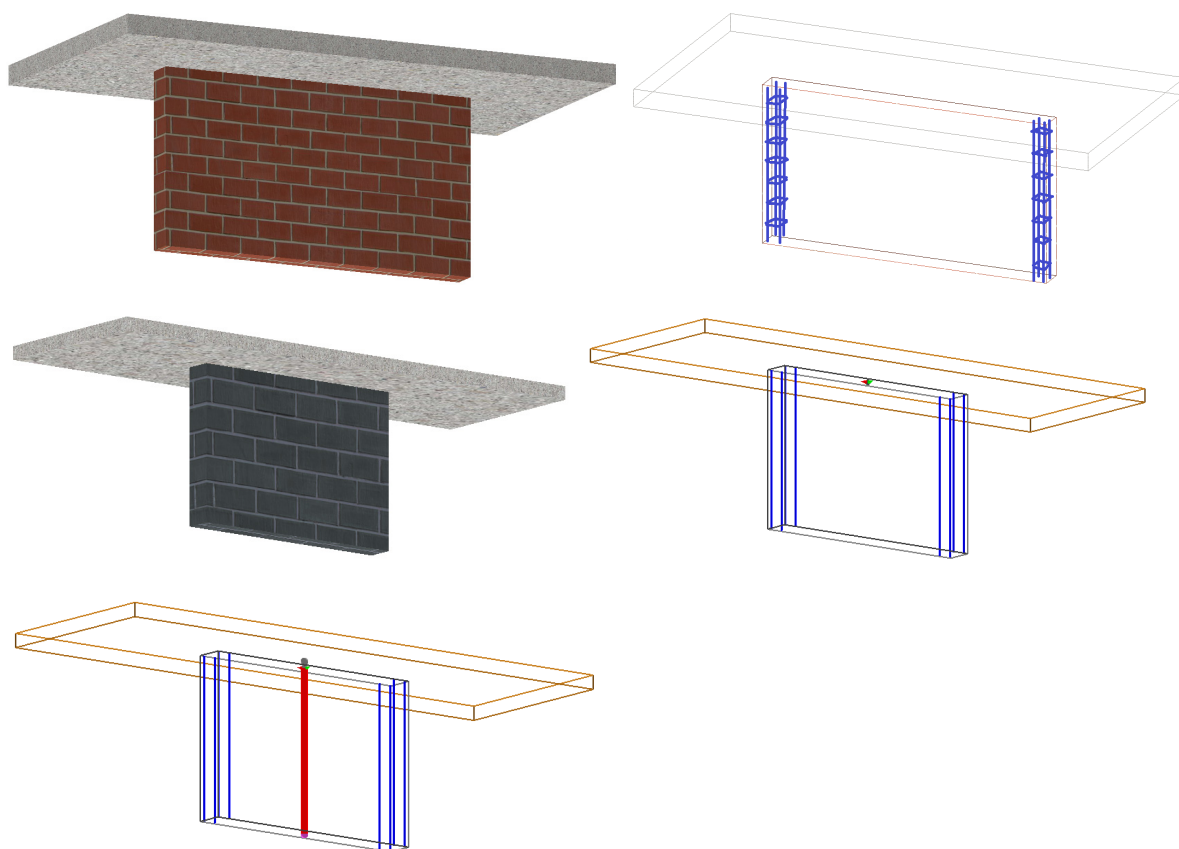


Figure 76: Model RM - modelling of confined masonry using a single wall with edge reinforcement

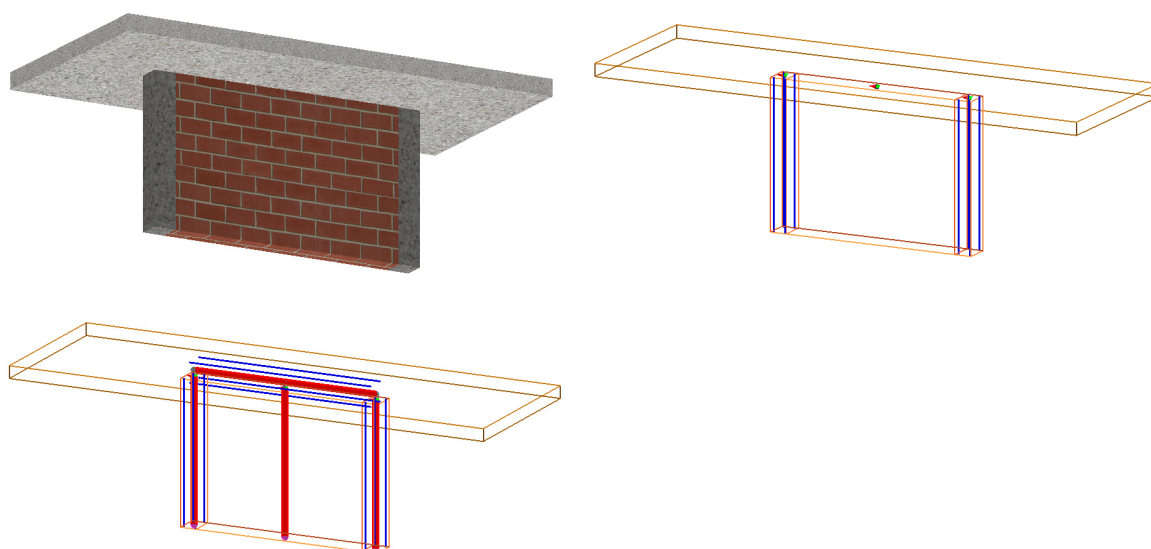
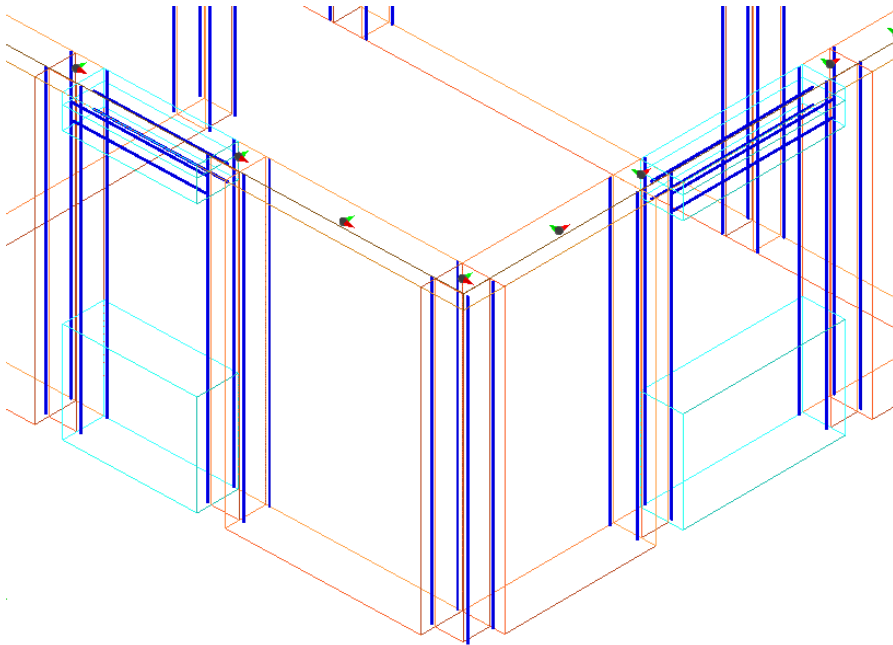
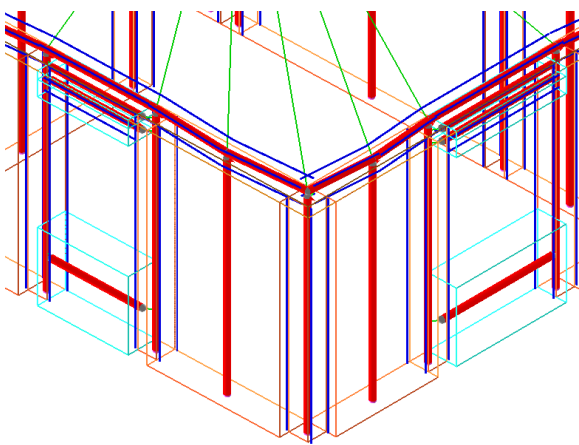


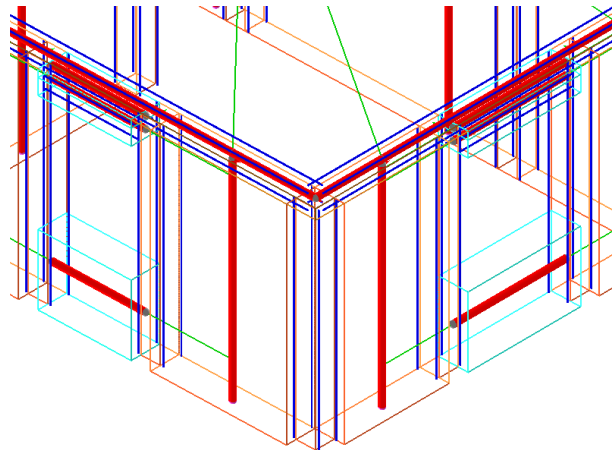
Figure 77: Model CM - modelling of confined masonry by two columns and one middle wall. The columns can be assigned either a reinforced masonry material or reinforced concrete material.



(a)



(b)



(c)

Figure 78: Automatic conversion from CM to RM model, (a) original model, (b) CM model, (c) automatically converted RM model. The automatic conversion is activated by the parameter **Change PM and RC to RM in the **Advanced Settings** of the **Mesh** dialog.**

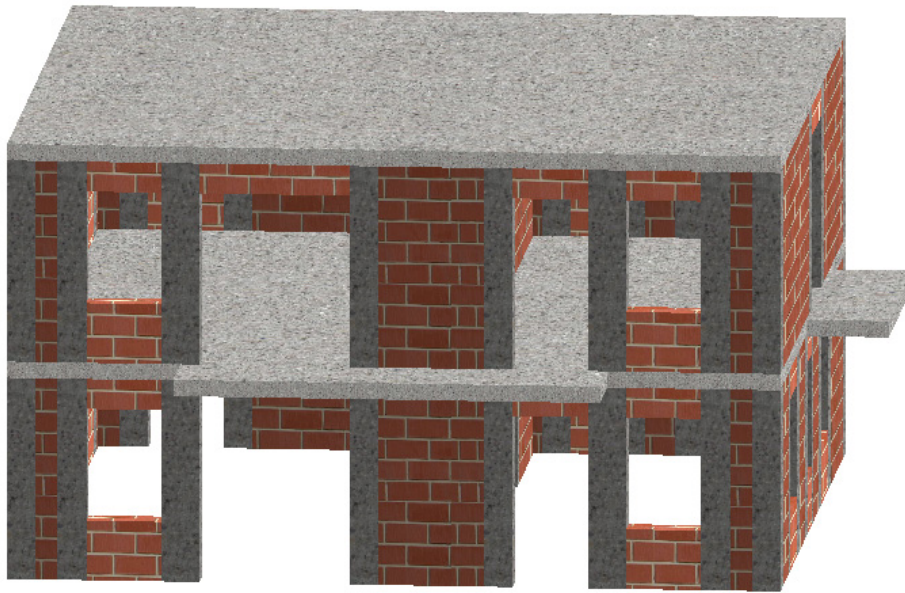


Figure 79: A model of a confined masonry building, which using the confined masonry model according to the method (c, i.e. Model CM)

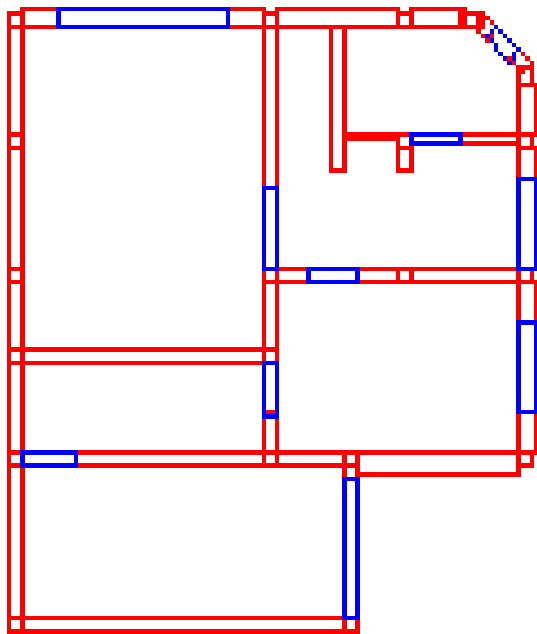
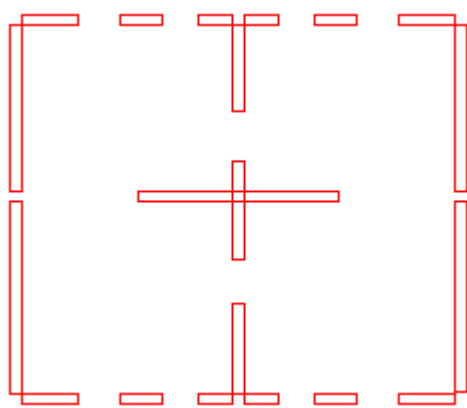
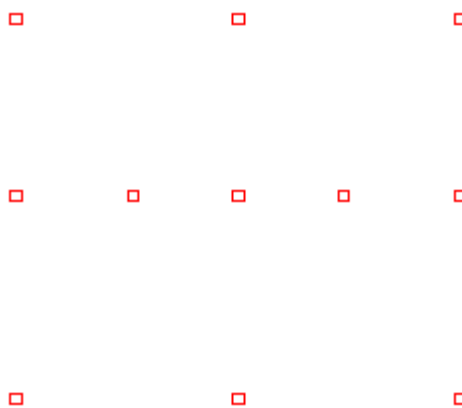


Figure 80: DXF file with the definition of confined regions for the confined masonry building



(a) layer for masonry walls



(b) layer for confining elements

Figure 81: DXF file definition with the separate layers for masonry elements and confining elements.

For the modelling of confined masonry it is possible to have all the wall elements in the same layer as it is indicated in Figure 80. With this method it is necessary to use the **Apply** button in the DXF import dialog in order to assign the correct material to the appropriate entities. This may be however quite tedious, so an alternative method may be considered.

An alternative approach for the confined masonry is to separate the definition of the walls into two layers: one for the masonry elements and one layer for the confining reinforced concrete columns. In this case, for each floor it will be necessary to perform two DXF imports. In the first one, the masonry walls, windows and ceiling is imported, and during the second import only the confining elements are imported.

4.3 How to model reinforced masonry

Reinforced masonry buildings are defined analogically to unreinforced masonry, but reinforced masonry material should be used instead. This material assumes that special units are used, which includes internal openings for inserting the vertical or horizontal reinforcement and a concrete infill.

The vertical reinforcement is defined by a number of reinforcement rows and spacing. Different spacing can be defined for the wall edges and for the wall middle part.

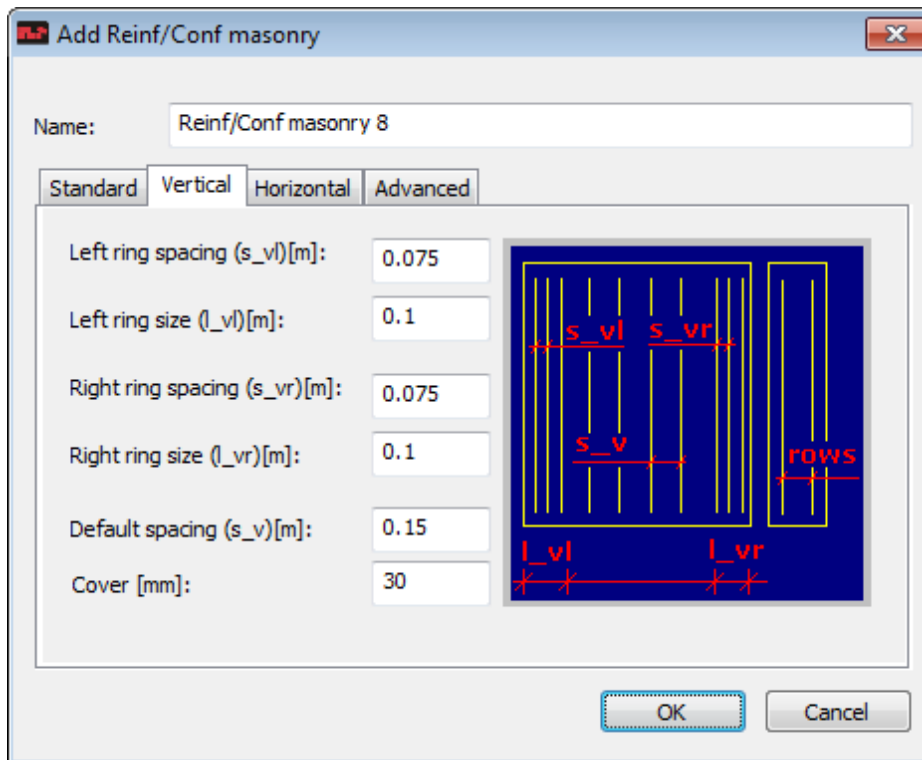


Figure 82: Definition of vertical reinforcement for reinforced masonry

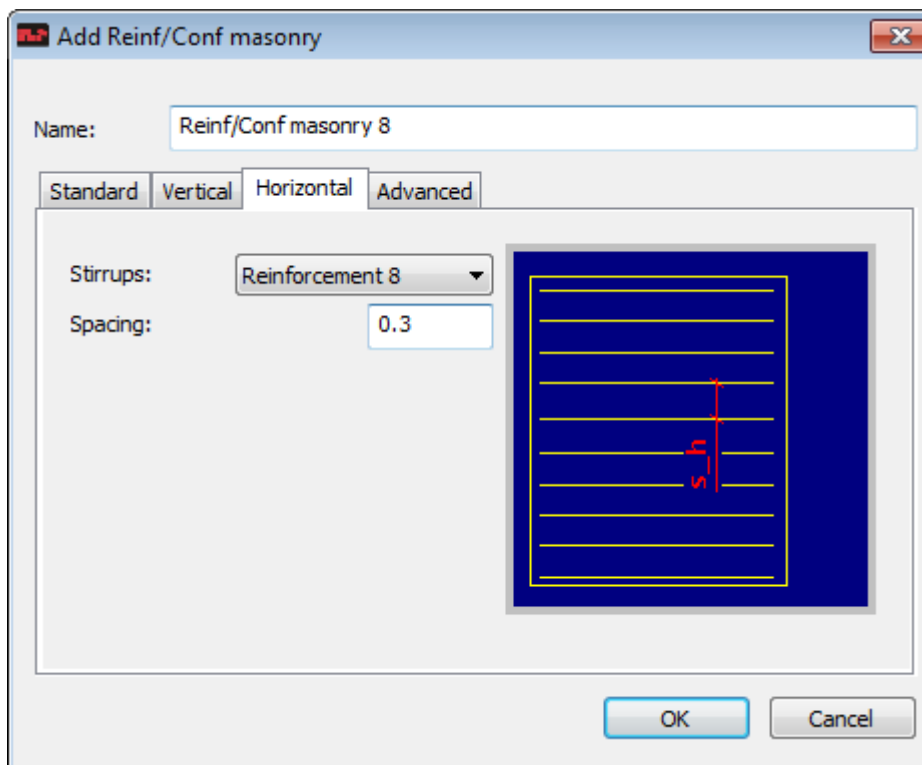


Figure 83: Definition of horizontal reinforcement for reinforced concrete masonry

4.4 How to model infill masonry

The infilled masonry can be modelled in a very similar manner to the confined masonry. During the definition of the DXF file, it is necessary to indicate, where the reinforced columns should be placed. These columns should be assigned a reinforced concrete material with the

appropriate reinforcement arrangement. The masonry walls in between shall be assigned an unreinforced masonry material. When defining this unreinforced masonry material, the check box for infill masonry should be selected. This will activate specific generation of the finite element model. In this model, the elements with in-filled masonry will not be inserted. Only their weight will be applied into the appropriate finite element nodes in the model.

4.5 How to model reinforced concrete elements

Reinforced concrete elements are modelled as normal walls or columns. The only difference is that reinforced concrete material should be assigned to these elements. In this case, the ultimate forces, i.e. the section strength in flexure or shear is calculated according to the Eurocode 2 formulas (see Section 5.3.2).

4.6 How to improve convergence and reduce errors

When convergence problems occur during the pushover analysis, the program will report high values of the solution error. An example is shown in Figure 84. After each analysis, the user should check whether the analysis errors remain in acceptable limits. When the error is above 10%, the results should not be used, and the analysis should be recalculated with different parameters. The convergence characteristics of the solution could be improved by modifying the settings in the dialog **Calculate | Advanced Settings**.

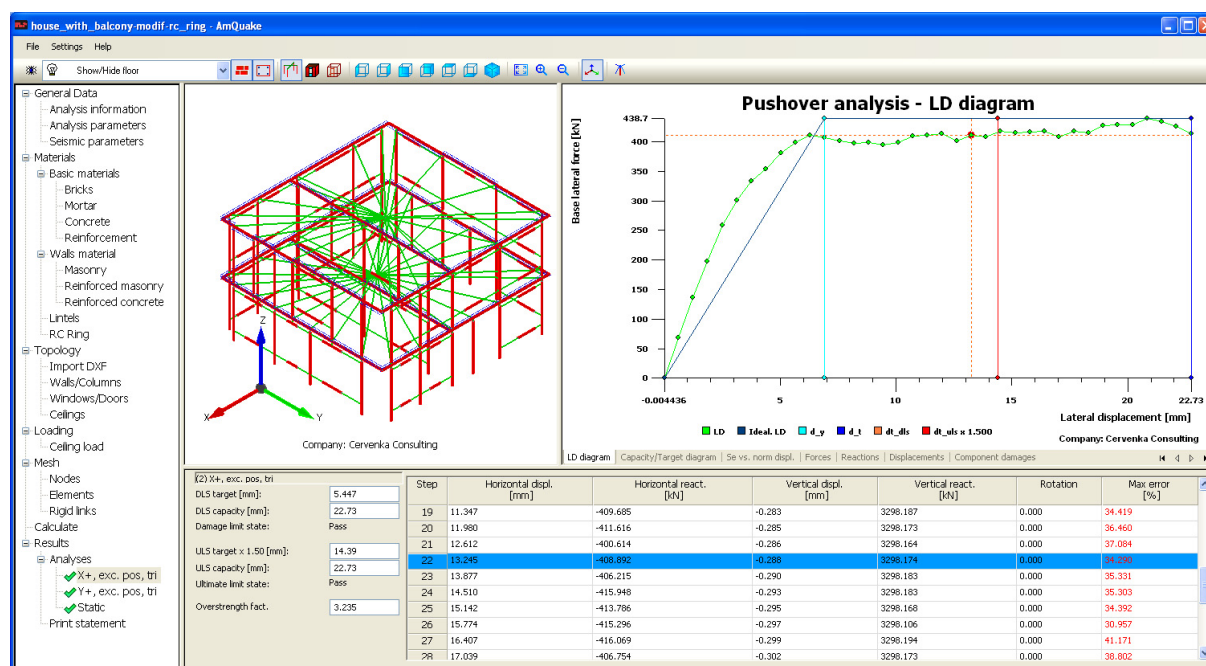


Figure 84: Analysis with convergence problems and high errors

In this case it is recommended to modify the following parameters:

Step length = activate the definition of this parameter and start reducing the default value, for instance by 50% reduction.

Max rel step length = if convergence problems persist, gradually reduce this parameter starting from 10 up to 1. However, this value should not drop below 1.

Min rel. step length = set this parameter to 1, if the analysis becomes too slow or too many steps are used to calculate the pushover curve.

4.7 How to model r.c. ring

Reinforced concrete ring will be created automatically for each ceiling when an r.c. ring material is selected in the **Ceilings** definition (see Figure 85). Elements for the reinforced concrete ring will be generated automatically when the mesh generation is executed.

In order to ensure that a continuous ring is created it is necessary to observe the following rules:

- if r.c. ring should be crossing over an open space between two walls, it is necessary to define a window between the two walls
- walls on each side of a window should have the same orientation of the local z-axis as the window between them (see Section 3.5).
- neighbouring walls that are to be connected by an r.c. ring should have the same orientation of their local z-axis. The correct definition of walls for a continuous r.c. ring generation is shown in Figure 86
- neighbouring walls that are to be connected by an r.c. ring should have the central points of their side faces close to each other. The correct definition is described in Figure 87.

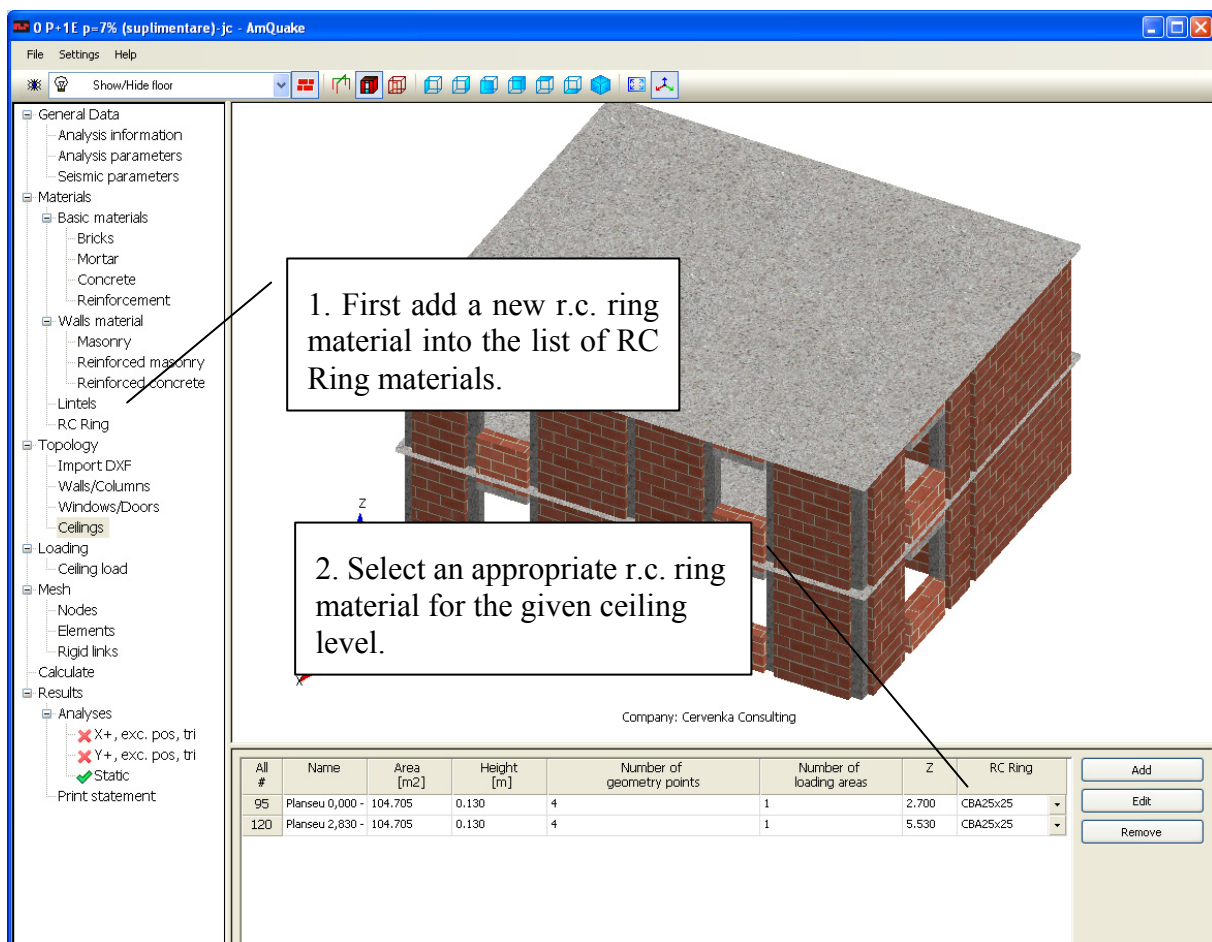


Figure 85: Reinforced concrete ring definition

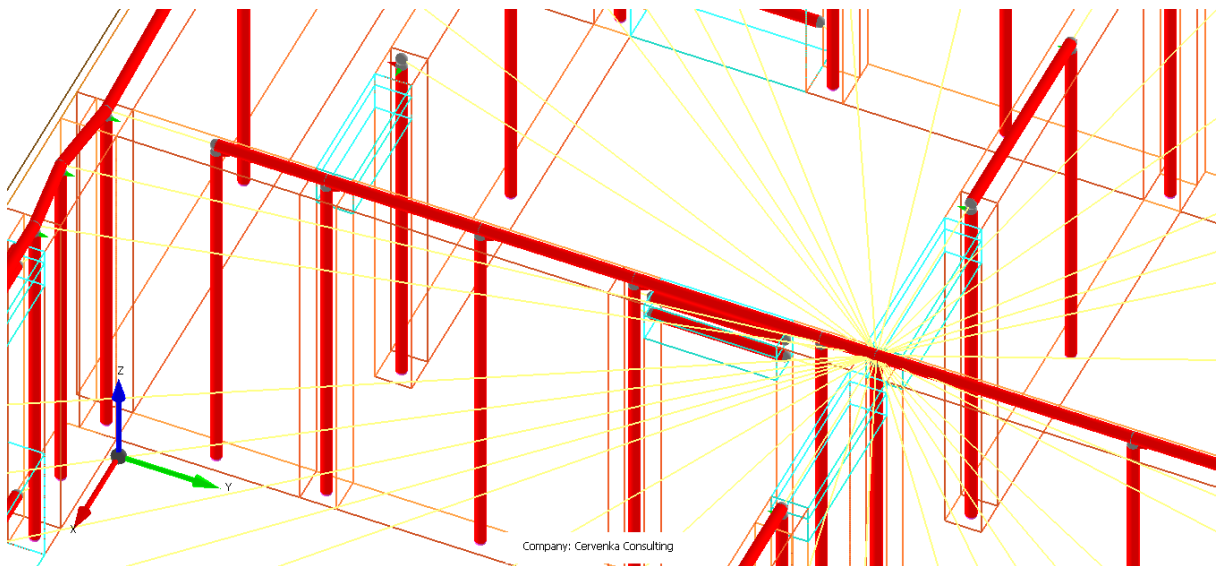
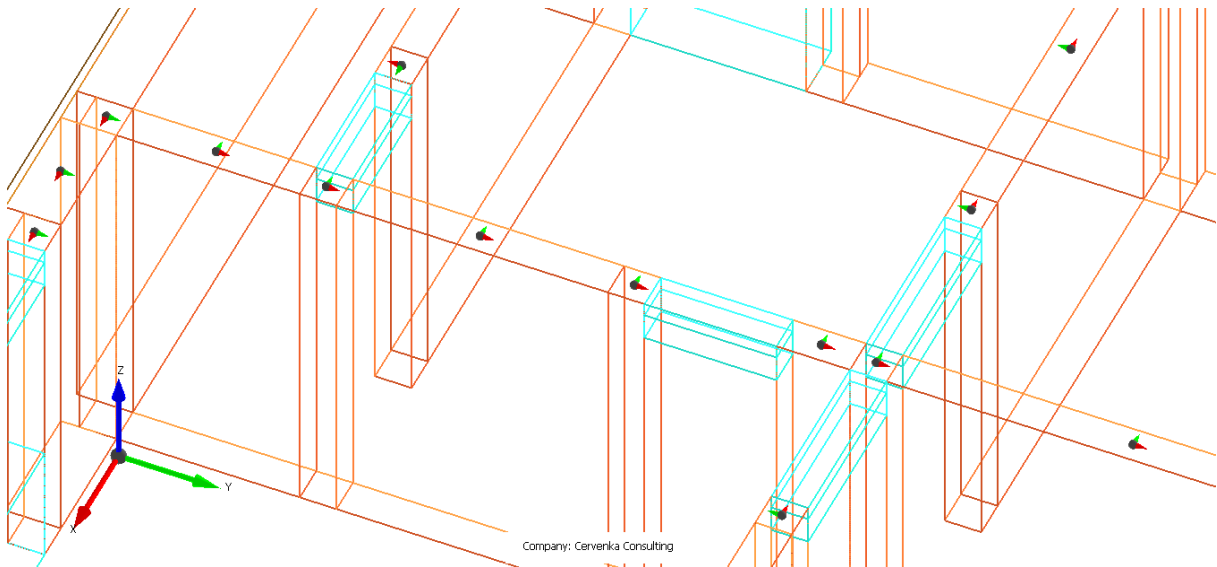
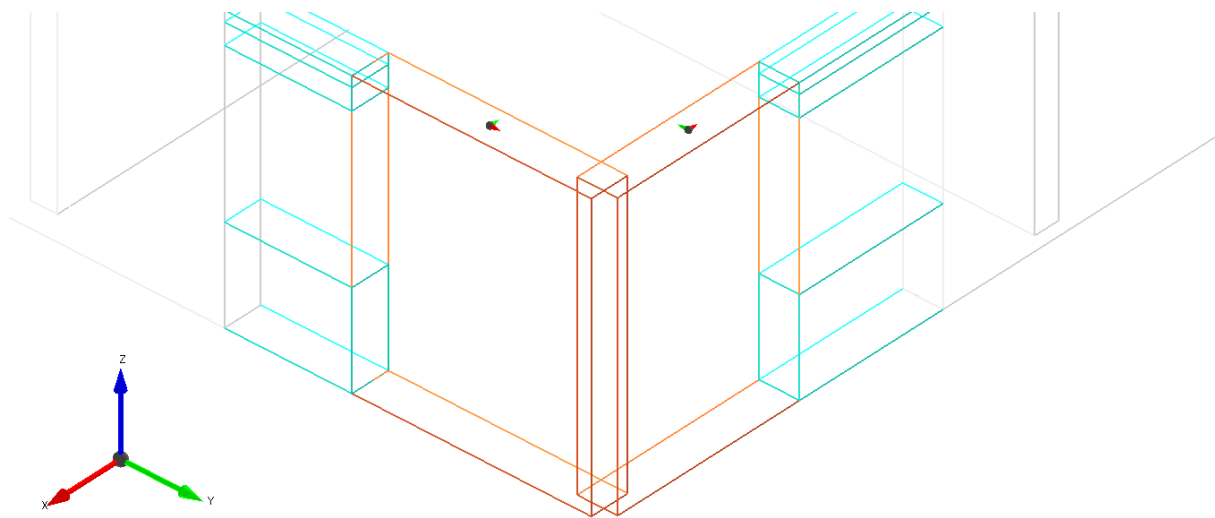
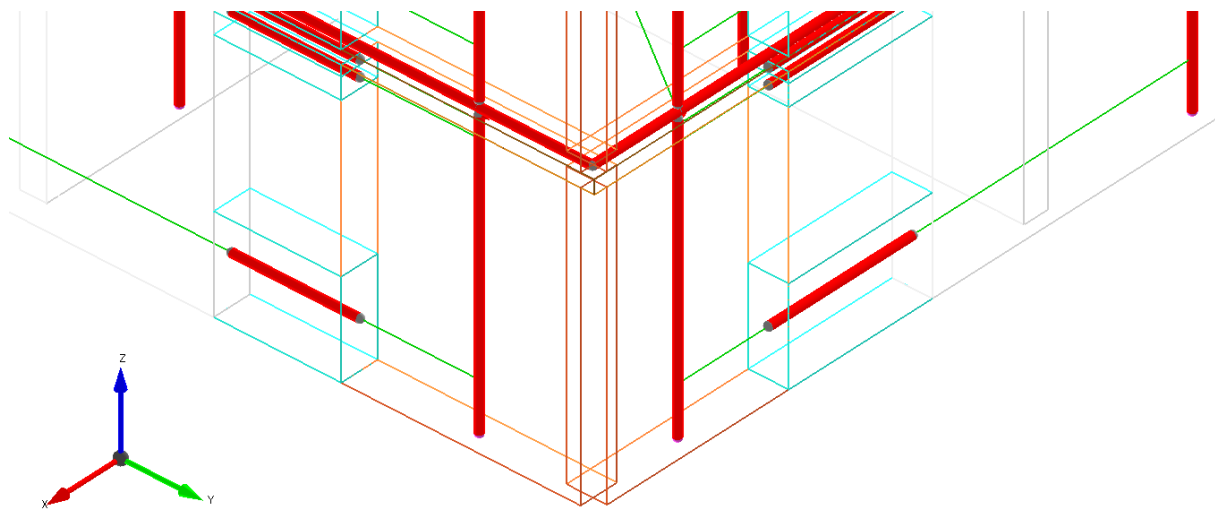


Figure 86: Continuous r.c. ring is generated for neighbouring walls and windows/doors with identical orientation of the local z axis.



Company: Cervenka Consulting



Company: Cervenka Consulting

Figure 87: R.C. ring is generated through the corner of the building if the corner walls are defined such that they slightly overlap each other, and the center of their side faces is close to each other.

5 THEORY MANUAL

This chapter includes the theory description of program **AmQuake**. The seismic assessment is performed according to Eurocode 8 and 6 using the pushover analysis and equivalent frame method.

In addition the program supports also a static analysis of masonry building according to the Eurocode 6.

AmQuake program includes national specifics as they are defined in corresponding annexes to EC6 and EC8 for the supported countries. In addition the program supports Romanian standards P100 and CR6 (2006 and 2013). The theory manual is written in general terms that should be applicable for all implemented national norms. Various national standards are activated by suitable choice of the individual parameters in the reported formulas. If a special treatment is required by some national implementation, it is indicated in the text.

The national specific parameters for the various formulas implemented in the program are defined by the national coefficient table, which is printed in Appendix A of this document.

5.1 Pushover Analysis

The conditions and method of the pushover analysis are described in EN 1998-1 or for Romania in P100. It is considered to be a nonlinear analysis with constant gravity loads and monotonically increasing horizontal loads. It maybe used to verify the structural performance of newly designed and of existing buildings as an alternative to the design based on linear-elastic analysis using the behaviour factor q .

The basis of the pushover analysis is to check the structural displacement with respect to the displacement demand based on an equivalent SDOF response using the elastic response spectrum $S_e(T)$ (EN 1998-1, 3.2.2).

$$0 \leq T \leq T_B : S_e(T) = a_g \cdot S \left[1 + \frac{T}{T_B} (\beta_0 \eta - 1) \right]$$

$$T_B < T \leq T_C : S_e(T) = a_g \cdot S \cdot \beta_0 \cdot \eta$$

$$T_C < T \leq T_D : S_e(T) = a_g \cdot S \cdot \beta_0 \cdot \eta \cdot \frac{T_C}{T}$$

$$T_D < T \leq 4s : S_e(T) = a_g \cdot S \cdot \beta_0 \cdot \eta \cdot \frac{T_C T_D}{T^2}$$

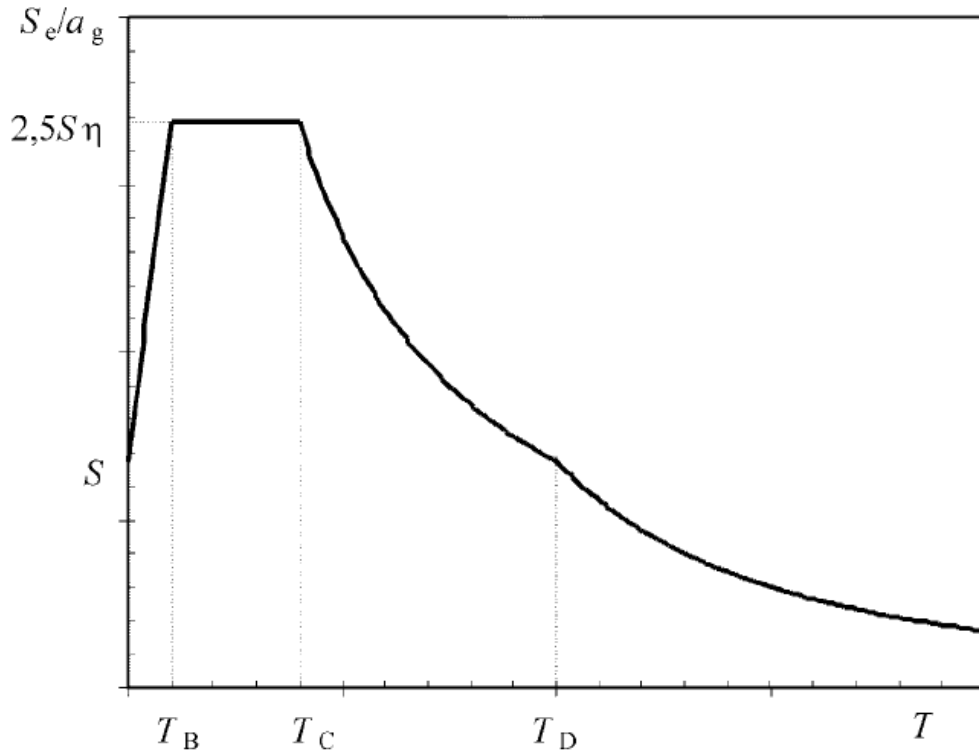


Figure 88: Shape of the elastic response spectrum

Parameters $a_{gR}, S, T_B, T_C, T_D, \beta_0$ depends on the national annex, soil conditions, ground type and spectrum Type Tables 3.2 and 3.3 in EN 1998-1 or other associated national standards.

In general the elastic response spectrum $S_e(T)$ is a function of the following quantities:

a_g - design ground acceleration on type A ground $a_g = \gamma_I a_{gR}$

γ_I - importance factor, topographic amplification factor for important structures (user input)

a_{gR} - reference peak ground acceleration on type A ground (country map, list of cities, user input)

T_B - lower limit of the period of the constant acceleration branch

T_C - upper limit of the period of the constant acceleration branch

T_D - the value defining the beginning of the constant displacement response range of the spectrum

S - soil factor (for Romanian P100 this factor is equal to 1.0)

η - damping correction factor, reference value of $\eta=1$ for 5% viscous damping, (for Romanian P100 this factor is equal to $\eta=0.88$ for $\xi=8\%$)

β_0 - dynamic amplification factor. Typical value in EN 1998-1 is 2.5, but may be different in various countries.

These quantities are given in EN 1998-1, 3.2.2.2 or national annexes for a given ground type A,B,C,D, E.

Ground types are defined in EN 1998-1, 3.1.2 or national annexes.

γ_I is defined in national annexes. In most cases equal to 1.0

a_{gR} is specified in national annexes and will depend on the seismic zones. It is specified for the two limit states:

ULS, i.e. ultimate limit state on the basis of the reference probability of exceedance P_{NCR} in 50 years or reference return period T_{NCR}

DLS, i.e. damage limit state on the basis of the “damage limitation requirement” probability of exceedance P_{DLR} in 10 years or a return period T_{DLR}

Ground acceleration a_{gR} comes from national maps

Acceleration for ultimate limit state

$$a_g = \gamma_I a_{gR}$$

Acceleration for damage limit state

$$a_{Dg} = \left(\frac{T_{LR}}{T_L} \right)^{-\frac{1}{3}} \gamma_I a_{gR} = \left(\frac{50}{10} \right)^{-\frac{1}{3}} \gamma_I a_{gR} = 0.584 \gamma_I a_{gR} = \gamma_D \gamma_I a_{gR}$$

γ_I is the importance factor from 3.3.2

γ_D is the damage state importance factor calculated as above.

5.1.1 EN 1998-1 assumptions for pushover analysis

1. Flexural and shear elastic stiffness of the masonry and reinforced concrete elements may be taken as ½ the original value to simulate the effect of cracking (EN 1998-1, 4.3.1(7))
2. As a minimum bi-linear force-deformation relationship should be used at the element level. (EN 1998-1, 4.3.3.4.1(2))
3. Zero post-yield stiffness maybe assumed. (EN 1998-1, 4.3.3.4.1(3))
4. If strength degradation is expected, it should be considered in the force-deformation relationship. (EN 1998-1, 4.3.3.4.1(3))
5. Mean values should be used for the material properties (EN 1998-1, 4.3.3.4.1(4))
6. Axial forces due to gravity shall be taken into account. EN 1998-1, 4.3.3.4.1(5).
7. Spatial model is necessary for irregular structures. Two analyses are necessary. One in each direction. It is not necessary to consider horizontal forces in two directions at the same time (EN 1996-1, 4.3.3.4.2.1 (2))
8. Two horizontal distribution of lateral forces should be applied
 - uniform, proportional to mass and independent of elevation
 - modal, EN 1998-1, 4.3.3.2 and 4.3.3.3, if

i. elevation regularity EN 1998-1, 4.2.3.3

ii. main period of vibration $T_1 \leq \begin{cases} 4T_C \\ 2s \end{cases}$

then lateral force for each storey

$$F_i = F_b \frac{z_i m_i}{\sum z_j m_j}$$

where

z_i - storey/mass elevation

m_i - storey mass according to EN 1998-1, 3.2.4

F_b - base shear

9. Lateral loads should be applied at the location of the masses, i.e. centres of the floors with eccentricity according to En 1998-1, 4.3.2(1).

$$e_{ai} = \pm 0.05 L_i$$

Romanian CR6-2013 code set this value to $e_{ai} = \max(L_i / 30, z_i / 300) \geq 1 \text{ cm}$

L_i - floor dimension perpendicular to the seismic action

Detailed description of how to calculate load eccentricity is beyond scope of this document. It is available in particular building codes, e.g. Eurocode 6, Section 6.1 and Annex G. Three column's/wall's cross sections are assessed: at the top, middle and bottom location. Calculation of the middle location is more complicated, because one usually have to account for wall slenderness, (i.e. buckling),too. (Note that this is not the case of Romanian CR6-2013).

10. Vertical component of the seismic action may be neglected in pushover analysis (EN 1996-1, 4.3.3.5.2)
11. Capacity curve is a relationship of base shear F_b versus top displacement, i.e. control displacement d_n .
12. Capacity curve should go from 0 to 150% the target displacement, EN 1998-1, 4.3.3.4.2.3
13. Target displacement is derived from the seismic demand from the elastic response spectrum in terms of a displacement of an equivalent SDOF system. Annex B, EN 1998-1
14. Capacity curve can be used also to determine

- Overstrength factor α_u / α_1 , both lateral distributions should be used

α_u - level of global plastic mechanism (failure level)

α_1 - level of first plastic hinge in the walls

- Plastic mechanism, both lateral distributions should be used and shall conform to the mechanism, on which the behavior factor q is used.

5.1.2 Target displacement

1. Capacity curve F_b / d_n must be converted into a capacity curve F^* / d^* of an equivalent SDOF system.

$$F^* = F_b / \Gamma_f \quad \text{and} \quad d^* = d_n / \Gamma_d$$

where

$$\text{for EC8:} \quad \Gamma_f = \Gamma_d = \frac{\sum \bar{F}_i}{\sum \left(\frac{\bar{F}_i^2}{m_i} \right)},$$

for P100:

$$\Gamma_d = \frac{\sum \bar{F}_i}{\sum \left(\frac{\bar{F}_i^2}{m_i} \right)}, \quad \Gamma_f = \frac{\sum \bar{F}_i}{\sum m_i} \Gamma_d$$

\bar{F}_i - normalized lateral load such that $\bar{F}_n = m_n$, where n indicated top floor.

$$\bar{F}_i = m_i \Phi_i, \quad \Phi_n = 1, \quad \text{where } n \text{ is the control node usually top floor}$$

$$m^* = \sum m_i \Phi_i = \sum \bar{F}_i \quad \text{for EC8}$$

$$m^* = \sum (m_i \delta_i^2) \quad \text{for P100}$$

2. This equivalent capacity curve is to be converted into an idealized elasto-plastic relationship.

F_y^* - ultimate strength

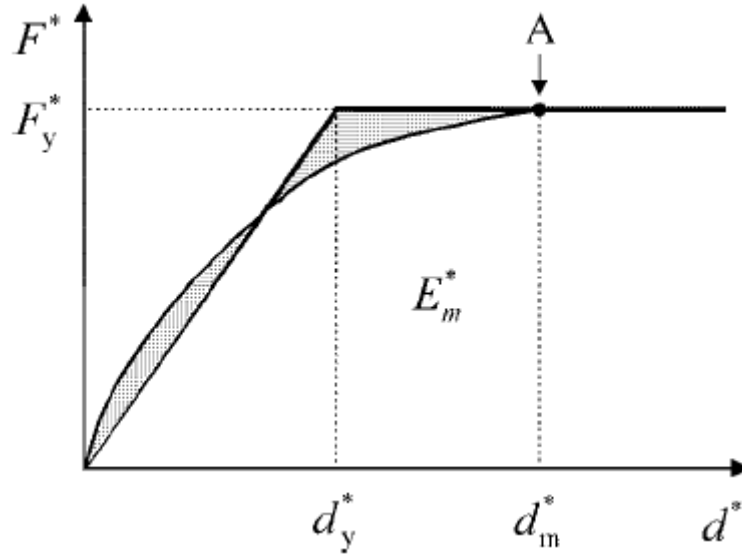


Figure 89: Definition of an idealized relationship

Assuming the preservation of energy under the equivalent capacity curve E_m^*

$$d_y^* = 2 \left(d_m^* - \frac{E_m^*}{F_y^*} \right)$$

3. Calculate period of the idealized SDOF system.

$$T^* = 2\pi \sqrt{\frac{m^* d_y^*}{F_y^*}} \text{ for EC8, } T^* = 2\pi \sqrt{\frac{m d_y^*}{F_y^*}} \text{ for P100}$$

4. Target displacement for an elastic behaviour

$$d_{et}^* = S_e(T^*) \left[\frac{T^*}{2\pi} \right]^2$$

5. Target displacement for a SDOF nonlinear system

For EC8:

if $T^* < T_C$ (short period range)

if $F_y^* / m^* \geq S_e(T^*)$ the response is elastic $\Rightarrow d_t^* = d_{et}^*$

else response is nonlinear $\Rightarrow d_t^* = \frac{d_{et}^*}{q_u} \left(1 + (q_u - 1) \frac{T_C}{T^*} \right) \geq d_{et}^*$,

$$\text{where } q_u = \frac{S_e(T^*) m^*}{F_y^*}$$

else (medium and long period)

$$d_t^* = d_{et}^*$$

For P100:

$$d_t^* = c d_{et}^*, \quad \text{where } 1 \leq c = 3 - 2.5 \frac{T^*}{T_c} \leq 2$$

6. Convert to MDOF target displacement

$$d_t = \Gamma_d d_t^*$$

Based on the above and the fact that $S_e(T)$ depends on the a_{gR} (i.e. reference peak ground acceleration), which is different for the two limit states ULS a DLS two values of d_t will be computed: d_t^{ULS} and d_t^{DLS} . First one is used to verify the ULS and the second for DLS.

Input needed for pushover analysis based on the above procedure from EN 1998-1:

Input	Description
Γ_f	Transformation factor between MDOF a SDOF for forces
Γ_d	Transformation factor between MDOF a SDOF for displacements
m^*	Equivalent mass of SDOF
$d(i)$	Displacement history of the control node at given dof
$F_b(i)$	History of the base shear
a_g	Design ground acceleration for No-collapse limit state NC, i.e. ultimate limit state
a_{Dg}	Ground acceleration for damage limit state DL, i.e. damage limitation state
S	Soil factor
T_B	Lower limit of the period of the period of the constant spectral acceleration branch
T_C	Lower limit of the period of the period of the constant spectral acceleration branch
T_D	Lower limit of the period of the period of the constant spectral acceleration branch
η	Damping correction factor, $\eta = 1$ for viscous damping of $\xi = 5\%$ $\eta = \sqrt{10/(5 + \xi)} \geq 0.55$
p_d	Relative value of displacement for stopping the analysis. Analysis is interrupted when the monitored displacement is larger than $d(n) \geq p_d \cdot d_t^{ULS}$ Default: 1.5
p_F	Relative value of shear force for stopping the analysis. Analysis is interrupted when the monitored shear force drops below $F_b \leq p_F \cdot \max(F_b(i))$ Default: 0,8

AmQuake monitors at each step i the deflections $d(i)$ and the base shear $F_b(i)$. At each step two values of target displacement d_t are calculated: d_t^{ULS} and d_t^{DLS} (i.e. ultimate state, damage state). They will be calculated using the procedure described earlier in this chapter based on the corresponding acceleration values:

a_g and a_{Dg} .

The analysis should continue as long as the last displacement $d(n)$ is larger than 150% of d_t^{UL} or when the base shear drops below 80% of the peak shear.

So analysis termination conditions are:

$$d(n) \geq p_d \cdot d_t^{ULS}$$

$$F_b(n) \leq p_F \cdot \max(F(i))$$

d_t^{DLS} will be only reported and used in the post-processing.

Seismic verification:

ULS verification:

$d(n) \geq p_d d_t^{ULS}$, i.e. the monitored deflection, i.e. the top storey deflection, at the last step (this step should satisfy the above two inequalities) should be larger than the required target displacement for ULS.

DLS verification:

$d(n_{DLS}) \geq d_t^{DLS}$, i.e. the monitored deflection, i.e. the top storey deflection, at step n_{DLS} should be larger than the required target displacement for DLS. The step n_{DLS} is determined such that it is the first step when the inter-storey drift at reaches the limiting value

$$d_r \leq 0.005 h,$$

where h is the storey height, the coefficient 0,005 is not a constant but depends on the definitions of Damage limit DLR in Section 3.3.2.

5.2 Equivalent frame method

The main principles of the equivalent frame method are described Reference [1]. The main idea of this approach lies in modelling all structural elements by 1D beam elements only. It applies for both vertical and horizontal elements like vertical bearing walls, horizontal bond/ring beams, door/window top heads etc. The next important assumption is that in-plane stiffness of all ceiling slabs is infinite, whilst their out-of plane stiffness is negligible. The solution is of predictor-corrector type. The predictor is based on purely linear elastic solution, whereas the corrector is nonlinear and obeys recommendations of Eurocode 6: Design of masonry structures. Similar approach is used also for reinforced concrete elements, but is based on Eurocode 2.

5.2.1 Finite element model

Implementation of the above model requires an implementation of quality finite element and suitable modelling of rigid body connections between the beams.

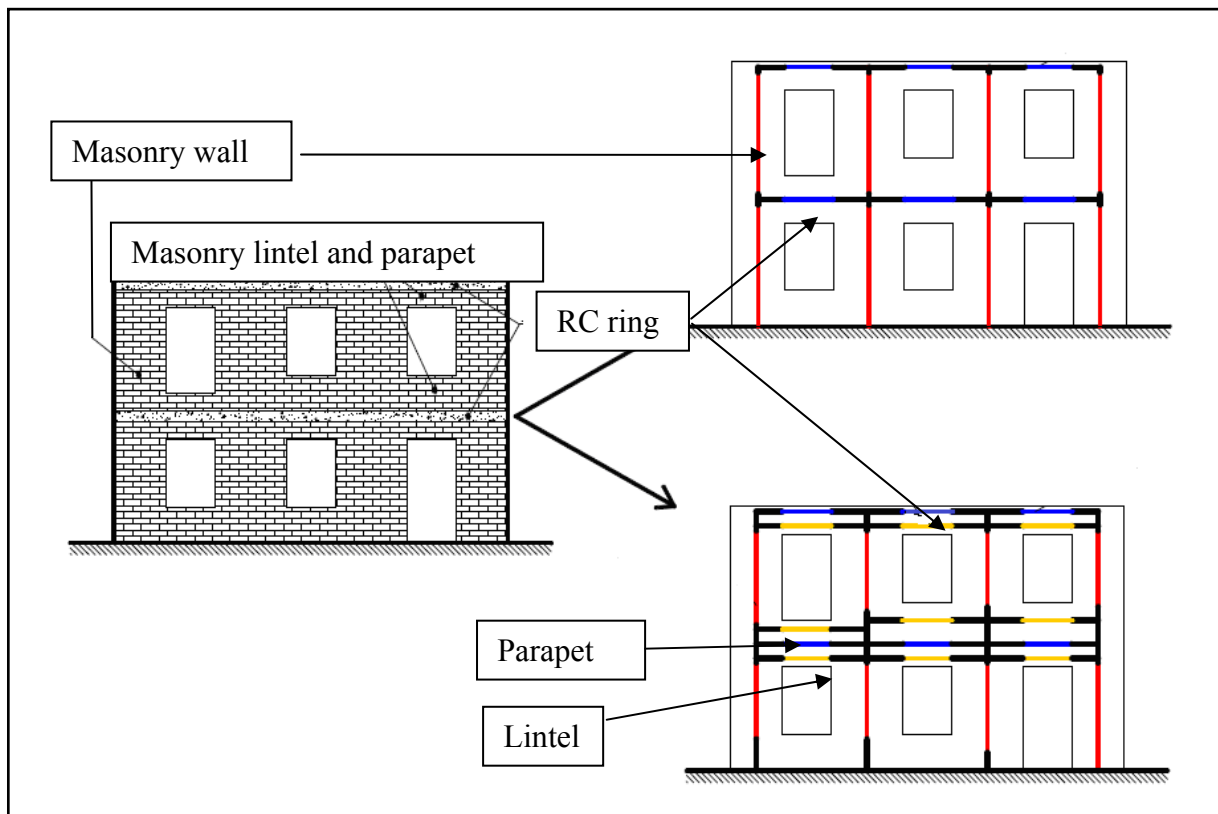


Figure 90: Structural model of a masonry wall disregarding (top) and considering (bottom) masonry lintels or parapets (Reference [1])

Figure of a typical bearing wall is depicted in Figure 90. Whether we account for door/window top heads or not, the beams can be both deep and slim. Hence, the beam finite element must incorporate accurate shear behaviour as well as bending behaviour. To meet all the above requirements, Timoshenko beam [6] is used.

Timoshenko element is based on cubic approximations of displacements. Rotations are calculated as derivatives of displacements. Unlike Kirchhoff element, Timoshenko beam

accounts also for shear strains γ_{xz}, γ_{yz} . The penalty for accurate approximation of deformations and good shear behaviour is a bit more complex formulation and higher computational cost.

5.2.2 Rigid body connections

A typical model consists of a number of bearing walls, walls with windows, RC rings, ceilings etc. With exception of ceilings, these are modelled by vertical or horizontal beam elements. Vertical elements are used to model bearing walls, while horizontal elements are used for parapet and lintel parts of walls with a window. The beams are typically placed to coincide with the axis of the corresponding (original) 3D solid element. Now, the question is how to interconnect the created beam elements, so that they behaviour is closed to behaviour of the original structure. In AmQuake, this is done by special boundary conditions that are described in this chapter.

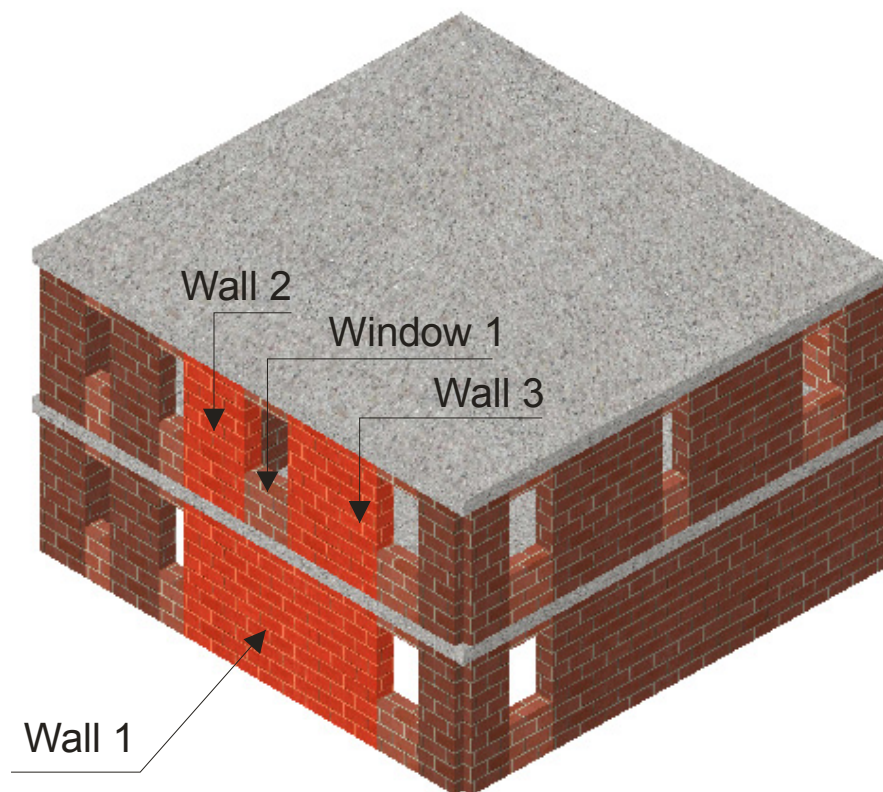


Figure 91: A 3D model of structure

The process of connecting individual walls and windows is demonstrated using the above figure. Let us assume modelling and mutual connection of Wall1, Wall2 and Window 1. They are model be beam elements. Details of Wall 1,2,3 and Window 1 is shown in the following figure:

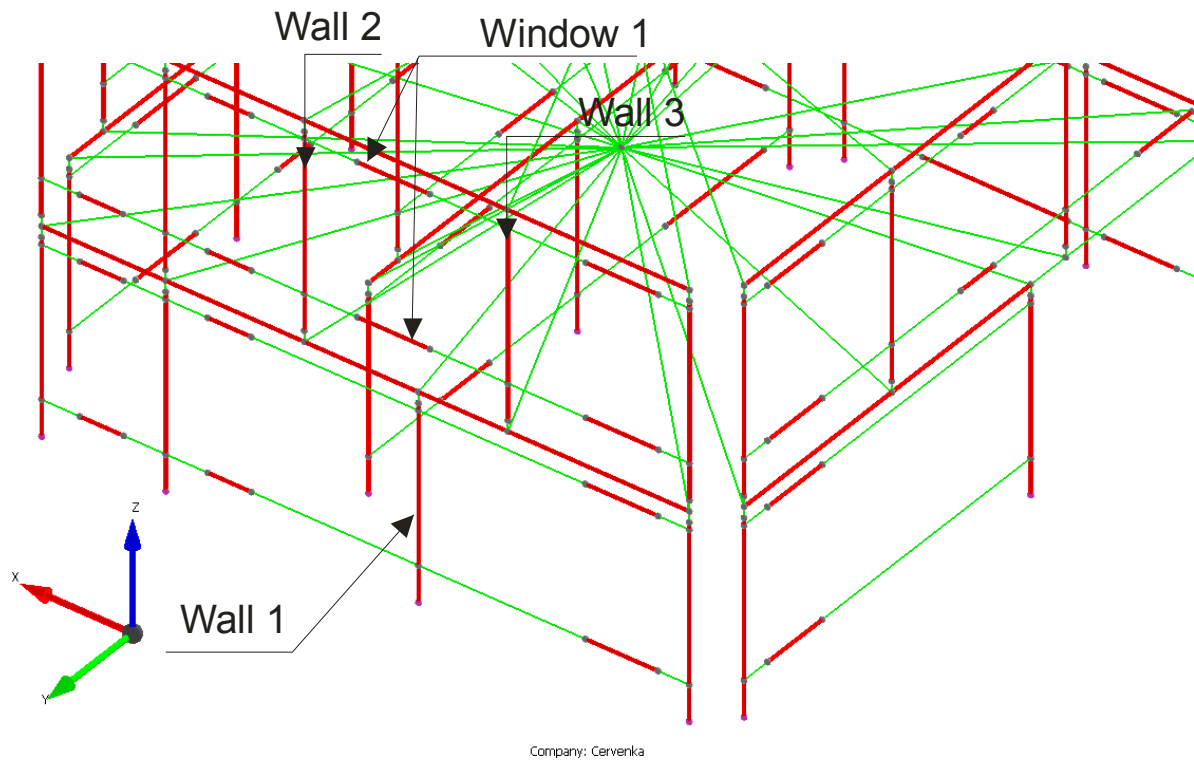


Figure 92: Wire model of a structure

The red lines in the picture represent beam elements and green lines indicate links between them. Bearing walls are modelled by vertical beams. Each wall with window is typically model by two beams, one for parapet and one for lintel part. Horizontal beams in the model are also used for RC rings at ceilings.

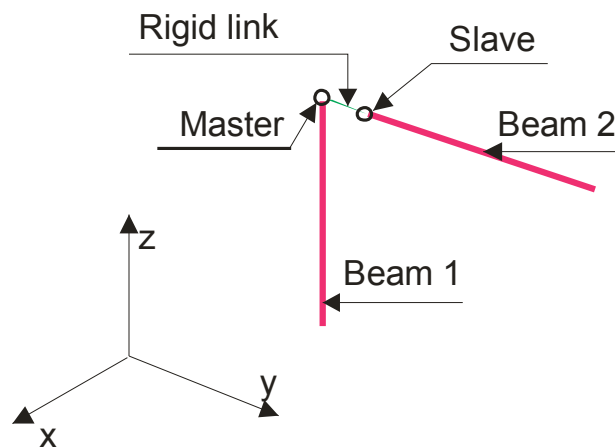


Figure 93: Rigid link

A special boundary condition is developed to mutually connect beams' end nodes. It is called RigidLink. Let us connect the slave node of Beam 2 to the master node of Beam 1, (see the above figure). It means that all displacements and rotations of the slave depend on related displacements and rotations of the master node. Physically one can imagine that the slave and the master nodes are connected by a very rigid rod. There are basically two ways to model such a rod:

1. Rigid body is represented by another beam with extremely high rigidity,
2. Using kinematic constraints.

The first option is used in [1]. It is a simple solution that does not involve anything new in the framework of a typical finite element program. It is often called solution by penalty method. However, as stiffness of the rigid parts of the structure must be much higher than stiffness of the other (“real”) beams, such solutions often suffers from rank deficiency. Hence, we are using the second approach: The rigid parts are modelled by Dirichlet boundary conditions, which are special case of the Multipoint constraints [6]. Mathematically these conditions are described in Equation (1.10):

$$\begin{aligned}
u_x^s &= u_x^m - \phi_y^m \Delta z + \phi_z^m \Delta y \\
u_y^s &= u_y^m - \phi_z^m \Delta x + \phi_x^m \Delta z \\
u_z^s &= u_z^m - \phi_x^m \Delta y + \phi_y^m \Delta x \\
\phi_x^s &= \phi_x^m \\
\phi_y^s &= \phi_y^m \\
\phi_z^s &= \phi_z^m
\end{aligned} \tag{1.10}$$

where $[u_x^s, u_y^s, u_z^s, \phi_x^s, \phi_y^s, \phi_z^s]$ are 6 global DOFS at the slave node, i.e. displacements and 3 rotations, $[u_x^m, u_y^m, u_z^m, \phi_x^m, \phi_y^m, \phi_z^m]$ are similar DOFS at the master nodes. The rotations ϕ are rotations along the axis indicated in the subscript and they are positive, if rotating anti-clockwise while watching opposite the axis. The slave and master nodes have coordinates $[x_s, y_s, z_s], [x_m, y_m, z_m]$, thus

$$\begin{aligned}
\Delta x &= x_m - x_s \\
\Delta y &= y_m - y_s \\
\Delta z &= z_m - z_s
\end{aligned} \tag{1.11}$$

Equation (1.10) expresses situation, when all DOFs of a slave are linked to all DOFs of a master node. This is the case of connection between upper end node of beams simulating bearing walls and a node atop the beam at the ceiling, connection of a parapet beam connected to an adjacent bearing walls etc.

Sometimes we apply the equations from (1.10) only for some DOFs. This is the case of ceiling slabs. The slabs have very high horizontal and negligible vertical rigidity. Hence, we need to link only horizontal DOFs. Equation (1.10) then reduces to

$$\begin{aligned}
u_x^s &= u_x^m - \phi_y^m \Delta z + \phi_z^m \Delta y \\
u_y^s &= u_y^m - \phi_z^m \Delta x + \phi_x^m \Delta z \\
\phi_x^s &= \phi_x^m
\end{aligned} \tag{1.12}$$

The above boundary conditions are generated automatically, (based on the beam nodal coordinates) and the slave DOFS are eliminated already during matrix assembly phase. As a results, the resulting linear governing equations comprise only “active” structural DOFS and there are no numerical problems due to penalty numbers.

In case of vertical beams, master nodes are typically placed atop and below each vertical beam at level of the central plane of the adjacent ceilings. These masters are then connected with slaves, i.e. end nodes of the beams themselves. For horizontal beams we also place slave nodes at both ends of the beams, however, the corresponding master nodes are located at intersection of those (horizontal) beams (to simulate parapets or lintels) and vertical beams,

(for bearing walls) that support them. Of course, in this case the vertical beams have to be split in two elements.

5.2.3 Connection between walls in adjacent floors

If bearing walls continue from one floor to the other, their mutual connection is simple. Their corresponding vertical beams are connected by the rigid link described above. However, how about the case, when the walls are not continuous between the floors?

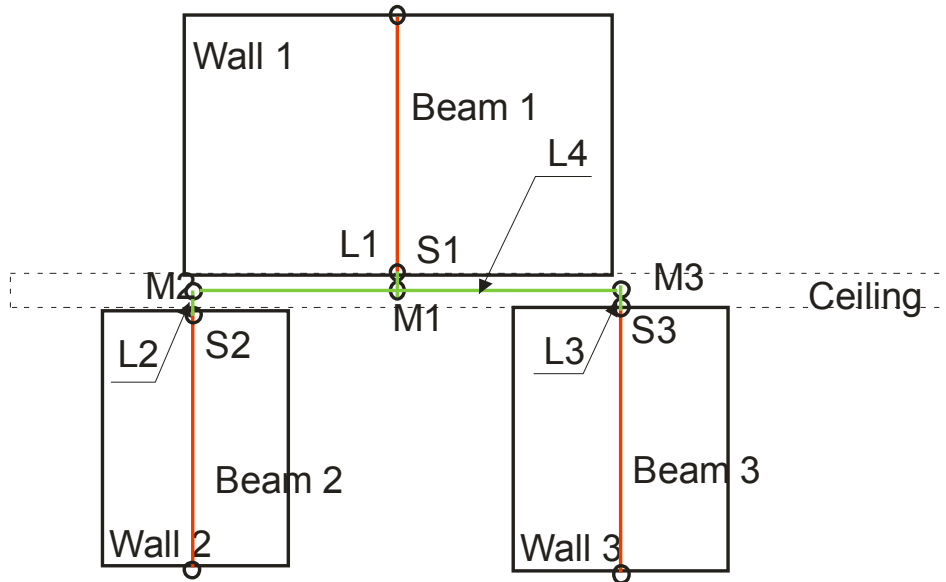


Figure 94: Vertical connection of walls

Let us connect Wall 1 from the upper floor to the walls Wall 2 and Wall 3 in the floor below as shown in Figure 94. Wall 1 is modelled by Beam 1, whose lower end node S1 is connected by usual rigid link to the node M1. Similar applies for walls Wall 2 and Wall 3 and corresponding nodes S2, S3, M2, M3.

As already noted, connection between nodes Si-Mi is simple, as 6DOFs of the slave node is connected by means of (1.10) to 6 DOFs of the master node. The three generated rigid links can be symbolically written as L1(M1,S1), L2(M2,S2) and L3(M3,S3).

A bit more complicated mutual connection is the link 4, i.e link between nodes M1, M2 and M3. Amquaque in this respect offers two options:

1/ Direct rigid link:

Two rigid links are applied: L4a(M1,M2) plus L4b(M1, M3). This option implicitly dictates L(M2,M3), or in other words all interface nodes from the lower walls are directly linked to the interface node from the upper wall. They all have the same rotations etc.

2/ Inverse rigid link:

One inverse rigid link is applied. Mathematically it reads:

$$\begin{aligned}
u_x^{M1} &= \varphi_2 \left(u_x^{M2} - \phi_y^{M2} (z_{M2} - z_{M1}) + \phi_z^m (y_{M2} - y_{M1}) \right) + \varphi_3 \left(u_x^{M3} - \phi_y^{M3} (z_{M3} - z_{M1}) + \phi_z^m (y_{M3} - y_{M1}) \right) \\
u_y^{M1} &= \varphi_2 \left(u_y^{M2} - \phi_z^{M2} (x_{M2} - x_{M1}) + \phi_x^m (z_{M2} - z_{M1}) \right) + \varphi_3 \left(u_y^{M3} - \phi_z^{M3} (x_{M3} - x_{M1}) + \phi_x^m (z_{M3} - z_{M1}) \right) \\
u_z^{M1} &= \varphi_2 \left(u_z^{M2} - \phi_x^{M2} (y_{M2} - y_{M1}) + \phi_y^m (x_{M2} - x_{M1}) \right) + \varphi_3 \left(u_z^{M3} - \phi_x^{M3} (y_{M3} - y_{M1}) + \phi_y^m (x_{M3} - x_{M1}) \right) \\
\phi_x^{M1} &= \varphi_2 \phi_x^{M2} + \varphi_3 \phi_x^{M3} \\
\phi_y^{M1} &= \varphi_2 \phi_y^{M2} + \varphi_3 \phi_y^{M3} \\
\phi_z^{M1} &= \varphi_2 \phi_z^{M2} + \varphi_3 \phi_z^{M3}
\end{aligned} \tag{1.13}$$

$$\varphi_2 = \frac{A_{1-2}}{A_{1-2} + A_{1-3}}$$

$$\varphi_3 = \frac{A_{1-3}}{A_{1-2} + A_{1-3}}$$

A_{i-j} denotes intersection area of the wall i and the wall j . Notice that nodes M2 and M3 are not anymore mutually constrained in this case. In practice it means that M1 is linked with M2 and M3 in a weighted form, where the applied weights depend on intersection area between the upper and lower wall.

Direct rigid links is good choice for walls lying in the same plane with large intersection area. Inverse rigid links present better option for cases with lateral supporting walls and for cases, when the upper wall has low stiffness or it is just a frame (and not a wall). In these cases fixation of rotational DOFs of all supporting beams would induce unrealistic stiffening of the whole structure.

Vertical walls' connection in AmQuake was demonstrated on the case of one upper wall supported by two walls underneath. A more complicated situation may arise. Nevertheless, it is treated in exactly the same way. Generally, the following rule applies:

Loop over all walls in the model and check their vertical support.

- 1/ If the wall is ground supported, no action is need. Proceed to the next wall.
- 2/ Otherwise find all walls from the floor below being in contact with the current wall. If such a wall (or walls) exist, connect them by Equation (1.10) or (1.13) and proceed to the next wall.
- 3/ If it does not exist, try to find a lintel(s) and/or RC ring(s) below the current wall. If it is successful, split the detected ring(s) and/or lintel(s) in two parts and connect the node at the splitting with the current wall. Again, this is done either by (1.10) or (1.13).
- 4/ In rarely cases it is not possible to find a wall or window below the current wall. The last chance is to find a loading area that covers location just below the current wall, (i.e. the node M1). If successful, the upper wall is linked with the wall corresponding to the detected load area.
- 5/ If all the above fail, the wall may only rely on supporting by some lateral frames.

It should be noted that the adopted Equivalent frame method is a tremendous simplification of a real 3D structure. Hence, the here applied links between the individual elements of the model also do not abound with accuracy. This is the penalty for simplicity and fast response in the presented **AmQuake**.

5.3 Nonlinear behaviour

This section brings description of nonlinear behaviour of beam elements simulating walls, lintels or columns.

5.3.1 Masonry

National catalogues contain the following data for each Wienerberger masonry product

Table 3: Masonry unit data

Data label	Type	Description
Product name	string	Name of the product
Masonry group	integer	Masonry group 1,2,3,4
Brick strength f_b	real	Normalized mean compressive strength of a single brick
Dimension d	real	Brick dimension d (thickness)
Dimension l	real	Brick dimension l (length)
Dimension h	real	Brick dimension h (height)
Mass	real	Mass density for dead load calculation
Tongue and groove	integer	0 or 1, i.e. NO or YES
Precision ground	integer	0 or 1, i.e. NO or YES
Reinforcement	integer	0 or 1, i.e. NO or YES, i.e. if it can be used as a reinforced masonry
Confined	integer	0 or 1, i.e. NO or YES, i.e. if it can be used as a confined masonry
Figure	string	File name of a JPG file containing the picture of the masonry unit
Product sheet	string	File name of a PDF file containing the product sheet for the given unit
Wall strength f_k	real	Characteristic strength of the wall made with this product. If not given, should be calculated using the formulas below
Wall initial shear strength f_{vk0}	real	Characteristic strength of the wall made with this product. If not given, should be calculated using the formulas below
E	real	Elastic modulus of the wall made with this product. If not given should be calculated using the formulas below.
G	real	Elastic modulus of the wall made with this product. If not given should be calculated using the formulas below.

f_{xk1}	real	Flexural strength, parallel to bed joints
f_{xk2}	real	Flexural strength, perpendicular to bed joints

Table 4: Mortar Properties

Name	Lightweight	Compressive strength f_m [MPa]	Brick precision ground	Mass Kg/m3
M5 General purpose	NO	5	NO	---
M10 General purpose	NO	10	NO	---
M15 General purpose	NO	15	NO	---
M20 General purpose	NO	20	NO	---
M5 lightweight	YES	5	NO	600
M10 lightweight	YES	10	NO	600

If the wall material properties are not given explicitly in the material catalogue they are calculated using the following formulas

Masonry compressive strength:

For units with precision ground

$$f_k = K f_b^\alpha f_m^\beta$$

and for units with precision ground

$$f_k = K f_b^\alpha$$

K is taken from Table 3.3 EN 1996-1-1 or Romanian CR6 or appropriate national annex. Indices α, β are taken from the corresponding national annex.

Masonry shear strength is calculated as a minimum of the following expressions:

$$f_{vk} = \min(r f_{vko} + \phi_{vk} \sigma_d, f_{vlt} + \phi_{vlt} \sigma_d) \quad (1.14)$$

where

$r=0.5$ for tongue and groove head joints

$r=1.0$ otherwise

$$f_{vlt} = 0.045 f_b, \text{ for tongue and groove head joints}$$

$$= 0.065 f_b, \text{ otherwise}$$

(for certain national standards such as for instance Romania CR6, f_{vlt} is determined as $f_{vlt} = 0,9 \cdot 0,034 \cdot f_b$)

$$f_{vko} \text{ is taken from EN 1996-1-1, Table 3.4}$$

ϕ_{vk} represents the friction coefficient for masonry shear strength calculation. Typically it is equal to 0.4, and it is defined in the national coefficients.

ϕ_{vlt} represents the friction coefficient for the second formula in (1.14). In eurocode, it is typically equal to zero, but in Romanian CR6 is defined as $\phi_{vlt} = 0.9 \cdot 0.14$.

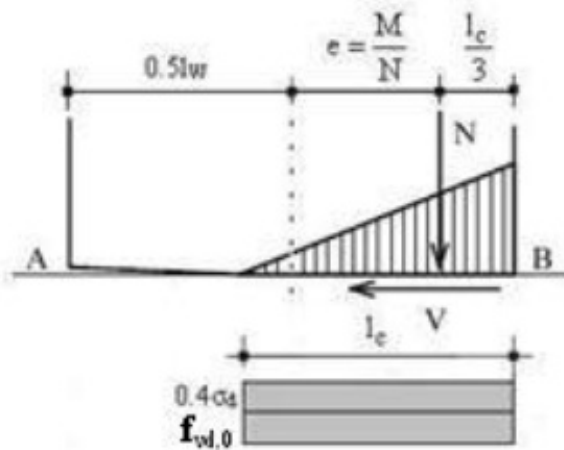
Romanian CR6-2013 operates with shear strength $f_{vk,l}$ and $f_{vk,i}$. The first strength corresponds to the above f_{vk} , but it is calculated in different way:

$$f_{vk,l} = f_{vk,0} + 0.4 \sigma_d \quad (1.15)$$

In AmQuake the coefficient 0.4 prior σ_d is computed using ϕ_{vk} / γ_m . Hence ϕ_{vk} must be input as $\phi_{vk} = 0.4 \gamma_m$, γ_m is safety factor. Moreover, value of σ_d is calculated as follows:

$$\begin{aligned} \sigma_d &= \frac{N_{Ed}}{t l_c} && \text{static load, confined \& unconfined wall} \\ \sigma_d &= \frac{N_{Ed}}{t l_w} && \text{seismic load, unconfined wall} \\ \sigma_d &= \frac{N_{Ed} + 0.8 V_{Ed} \frac{h_{pan}}{l_{pan}}}{t l_w} && \text{seismic load, confined wall} \end{aligned} \quad (1.16)$$

In the above expressions t, l_c, l_w stands for thickness, shear compressed and total length of the wall, respectively, h_{pan}, l_{pan} is height and length of the confined wall assemblage and N_{Ed} , V_{Ed} is designed normal compression and shear force, (both to be taken positive). $l_c = \frac{3}{2} l_w - 3e$, which is derived from:



The second shear strength accounts for diagonal failure due to tension effort for URM:

$$f_{vk,i} = 0.22 f_{bt} \sqrt{1 + 5 \frac{\sigma_{od}}{f_{bt}}} \quad (1.17)$$

where masonry strength $f_{bt} = 0.035 f_b$, f_b is characteristic normalized compressive strength of a masonry unit.

Masonry wall elastic Modulus E

$$E = K_E f_k$$

K_E is taken from EN 1996-1-1, 3.7.2

Masonry wall shear Modulus G

$$G = K_G E$$

K_G is taken from EN 1996-1-1, 3.7.3

Flexural strength f_{xk1} and f_{xk2} are not used in AmQuake as the out of plane bending is considered by constructing a general M-N interaction diagram for a bending in general direction. The out of plane bending perpendicular to the bend joints is also not considered in AmQuake since such a loading cannot be realized in AmQuake numerical model. If necessary such a special loading must be verified separately for the problematic wall.

The flexural strength of a masonry wall is determined using the same way as for reinforced concrete and is described in Section 5.3.3.

Shear strength assessments

$$V_{rd} = t l_c f_{vd} + r_h f_{yd} \frac{A_{sw}}{s} l_w + r_v f_{yd} A_{asc} \leq f_{yvd,max} t l \quad (1.18)$$

where:

- f_{vd}, f_{yd} is design masonry shear strength and design reinforcement strength
- A_{sw} is area of shear reinforcement in one horizontal joint.
- A_{asc} is area of vertical reinforcement in the compressed zone
- $f_{yvd,max}$ is maximum shear reinforcement strength being equal to 2MPa in case of CR6 this value is not used
- t wall thickness
- l_c area of compressed part considered as:
For EC6: $l_c = l$ if vertical reinforcement is present
For Romanian CR6:

$x_c \leq l_c = 3 \left(\frac{h}{2} - e \right) \leq l$, where e is the normal force eccentricity and x_c is the length of the compressed zone from the ultimate moment calculation in Section 5.3.3.

r_h coefficient for shear strength of horizontal reinforcement

$$r_h = 0.9 \quad \text{for EC6}$$

$$r_h = 0.8 \quad \text{for Romanian CR6}$$

l_w wall length

r_v coefficient for shear strength of vertical reinforcement

$$r_v = 0. \quad \text{for EC6}$$

$$r_k = 0.2 \quad \text{for Romanian CR6}$$

In Romanian code CR6-2013 Equ (1.18) is modified to:

$$V_{rd} = \min(V_{rd,l}, V_{rd,i}) \leq f_{yvd, \max} t l \quad (1.19)$$

where $V_{rd,l}$ resembles that from Equ (1.18) and $V_{rd,i}$ accounts for diagonal tension failure.

The shear strength $V_{rd,l}$ is computed by:

$$V_{rd,l} = t l_l f_{vd} + r_h f_{yd} \frac{A_{sw}}{s} l_w + r_v f_{yd} A_{asc} + V_{Rsc} \quad (1.20)$$

where r_h is participation factor of the vertical reinforcement of the tie-column, table 6.3. CR6-2014. The length l_l is:

$$l_l = l_c = 1.5 l_w - 3e \quad \text{static load}$$

$$l_l = l_{ad} = 2l_c - l_w \quad \text{seismic load}$$

$$l_{ad} = 2l_c - l_w \quad \text{I,U,T wall with cross section}$$

$$l_{ad} = \left(2.35 \frac{\sigma_d}{f_d} - 1 \right) l_w \quad \text{rectangular wall} \quad (1.21)$$

$$l_{ad} = \left(2.666 \frac{\sigma_d}{f_d} - 1 \right) l_w \quad \text{rectangular wall from Porotherm}$$

For seismic alternate action the adherence length stays active, if $\sigma_{od} \geq \frac{f_d}{2.67}$. Otherwise, $l_{ad} = 0$. The term V_{Rsc} accounts for the shear resistance of the concrete cross – section of the tie-column.

The shear strength $V_{rd,i}$ is computed by:

$$V_{Rd,i} = \frac{t l_w}{b} f_{vd,i} \quad (1.22)$$

where $b = \frac{h}{l_w}$, $b \in <1., 1.5>$

Finally, the forces V_{rd} are then compared with design shear forces $V_{Ed,y}, V_{Ed,z}$. When the ultimate resistance in shear is reached, the wall is assumed to behave as a perfectly plastic material until the limiting value of the shear drift is reached as described in Figure 95. The limiting values of shear drift are provided in the **Analysis parameters** (see Figure 18)

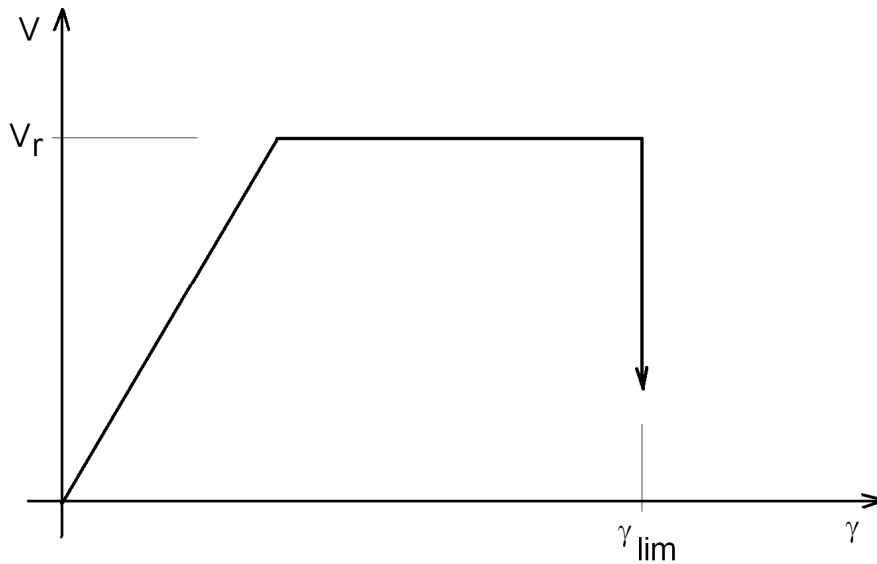


Figure 95: Idealized diagram for masonry wall shear behavior

5.3.2 Reinforced concrete

The reinforced concrete wall, column, lintel or RC ring elements are verified with respect to flexural and shear strength as described in Section 5.3.3 and 5.3.2.1 respectively.

5.3.2.1 Shear strength of RC element

The shear strength of an RC element is calculated according to Eurocode 2, i.e. EN 1992-1-1, sec. 6.2.2 and 6.2.3.

Concrete elements without shear reinforcement

$V_{Rd,c} = V_{Rd,cm} + V_{Rd,cn}$ (i.e. effect of concrete + effect of normal force)

$$V_{Rd,cm} = C_{Rd,c} k (100 \rho_1 f_{ck})^{1/3} b_w d$$

$$V_{Rd,cn} = k_I \sigma_{cp} b_w d$$

in addition the following condition has to be met

$$V_{Rd,cm} \geq v_{\min} b_w d$$

where:

$C_{Rd,c}$ - **input parameter**, based on National annex,

default value $0.18/\gamma_c$

$$k = 1 + \left(\frac{200}{d} \right)^{1/2} \leq 2.0, \text{ where } d \text{ is in mm}$$

$$\rho_1 = \frac{A_{s1}}{b_w d} \leq 0.02, \text{ effective reinforcement ratio}$$

f_{ck} **input parameter**, concrete characteristic compressive strength

b_w **input parameter**, element width, i.e. cross-section thickness

d **input parameter**, effective depth of the section, typically distance between the centre of the longitudinal reinforcement and the top edge.

A_{s1} **input parameter**, area of the longitudinal, bending reinforcement

k_I **input parameter**, typically 0.15

σ_{cp} average compressive stress, $\sigma_{cp} = \frac{N_{Ed}}{A_c}$, compression $N_{Ed} > 0$

A_c concrete area, should take into account reinforcement

v_{\min} **input parameter**, minimal shear strength,

default value $v_{\min} = 0.035 k^{3/2} f_{ck}^{1/2}$

Concrete elements with shear reinforcement

$$V_{Rd,s} = \frac{A_{sw} f_{ywd} z \cot \theta}{s}$$

but the following condition must be satisfied, (i.e. strength of the compressive strut)

$$V_{Rd,s} \leq V_{Rd,max} = \frac{\alpha_{cw} v_1 f_{cd} b_w z}{(\tan \theta + \cot \theta)}$$

where

$\cot \theta = (\alpha_{cw} v_1 \frac{f_{cd}}{v_{Rd,w}} - 1)^{1/2}$, θ is the shear crack angle, with the following condition

$$1 \leq \cot \theta \leq 2.5$$

$$v_{Rd,w} = \rho_w f_{ywd}$$

$$\rho_w = \frac{A_{sw}}{b_w s} \leq 0.5 v_1 \frac{f_{cd}}{f_{ywd}}$$

A_{sw} **input parameter**, area of reinforcement stirrups, (typically 2 x stirrup area)

f_{ywd} design yield strength of reinforcement stirrups, $f_{ywd} = f_{ywk} / \gamma_s$

f_{ywk} **input parameter**, reinforcement char. yield strength

γ_s **input parameter**, partial factor of safety for reinforcement, typically 1.15

z is the inner lever arm, for a member with constant depth, corresponding to the maximum bending moment in the element under consideration. In the shear analysis of reinforced concrete without axial force, the approximate value $z = 0,9d$ may normally be used

s stirrup spacing

α_{cw}	$(1 + \frac{\sigma_{cp}}{f_{cd}})$	for	$0 \leq \sigma_{cp} \leq 0.25 f_{cd}$
	1.25	for	$0.25 f_{cd} < \sigma_{cp} \leq 0.5 f_{cd}$
	$2.5(1 - \frac{\sigma_{cp}}{f_{cd}})$	for	$0.5 f_{cd} < \sigma_{cp} \leq 1.0 f_{cd}$

ν_1 **input parameter**, coefficient of compressive strut strength

0.6 for $f_{ck} \leq 60 \text{ MPa}$

$\left(0.9 - \frac{f_{ck}}{200}\right) \geq 0.5$ for $f_{ck} > 60 \text{ MPa}$

$f_{cd} = f_{ck} / \gamma_c$ design concrete compressive strength

f_{ck} **input parameter**, characteristic concrete compressive strength

γ_c **input parameter**, safety factor for concrete, typically 1.5

When the ultimate resistance in shear is reached, the element is assumed to behave as perfectly plastic materials until the limiting value of the shear drift is reached as described in Figure 96. The limiting values of shear drift is calculated using the same formula as for the flexural limit (see Section 5.3.3)

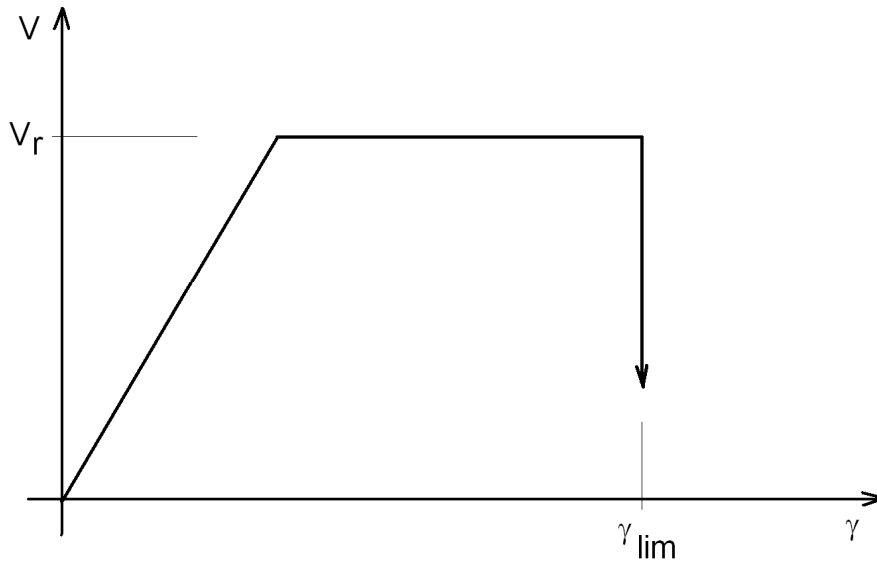


Figure 96: Idealized diagram for reinforced concrete shear behavior

5.3.3 Element strength evaluation

Walls, columns, lintels and RC rings are modelled in **AmQuake** by a beam finite element described in details in [11] and Section 5.2.1. It is a geometrically linear finite element based on beam theory, stating that during the loading the beam's cross sections remain always planar, although not necessarily normal to the beam's centre line. It introduces a special internal variable to account for tilting of its cross sections with respect to central line the element, which result in much better shear behaviour of the element. This approach is called Timoshenko theory, (or also Mindlin, Reissner theory). The element can be used to calculate shallow as well as deep beam structures, i.e. deep walls as well as shallow windows' lintels, RC rings etc. (Note that classical Kirchhoff element is suitable only for shallow structures).

The element has 6 DOFs at each of its ends, three displacements and three rotations. They are defined in global coordinate system. Internally, the element is computed in a local coordinate system, which is defined as follows:

Its longitudinal (local) axis x' coincides with the beam's central axis, (from the first to the second incidence node).

The second local axis, i.e. z' axis, is perpendicular to x' . Horizontal elements have this axis parallel with global z . For other elements, z' is located respectively in global x - z and y - z plane, depending on whether the rotation angle of the element (i.e. a parameter from input) is in range 0-45 or 135-225 or 315-360 degree, or it is out of the ranges.

The beam's deformations are outputted in global coordinate system. Strains and stresses are printed in the local coordinate systems.

The element uses cubical interpolation of displacements and independent quadratic interpolation of rotations along its length. For linear material and fixed end loading conditions the element provides exact solution, (deformations and internal forces for beams of any depth).

Masonry and/or concrete elements use nonlinear material law defined in [12] and the structures are solved iteratively as follows:

$$\Delta \underline{u}_i = \mathbf{K}_{i-1}^{-1} [\underline{Q} - \underline{F}_{i-1}] \quad (1.23)$$

$$\underline{u}_i = \underline{u}_{i-1} + \Delta \underline{u}_i \quad (1.24)$$

$$\underline{F}_i' = \mathbf{K}_i \underline{u}_i \quad (1.25)$$

$$\underline{F}_i = \text{check_MNT}(\underline{F}_i') \quad (1.26)$$

where \mathbf{K} is structural stiffness matrix, i relates to step/iteration number, \underline{u} is vector of displacements and \underline{F} and \underline{Q} are vector of internal and external nodal forces. Thus, in a new step/iteration we search for a new displacement increment Δu that minimizes unbalanced forces $\underline{Q} - \underline{F}_{i-1}$.

Predictor of the solution, i.e. the matrix \mathbf{K} is typically kept constant within a loading step. (Note that AmQuake can be asked to recalculate \mathbf{K} each iteration to account each iteration for changes in geometry and material conditions. For most case, however, it is not needed and by default the matrix \mathbf{K} is kept within each step constant).

As for the corrector, internal forces are firstly calculated using linear approach, i.e. using (1.25). Thereafter, AmQuake runs a loop through all beams to check, if the designed load does not exceed ultimate load, (see (1.26)). If it does, the beam's end nodal forces are appropriately reduced.

Note that such reduction must not violate element force/moment equilibrium. Hence reduction of a nodal force(s)/moment(s) due to Eurocode [12] is often accompanied by further reduction of ambient forces/moments to restore the beam's equilibrium.

The actual assessment of design vs. ultimate forces/moments is done according to appropriate Eurocode. It will be described here for the case of masonry reinforced walls. Reinforced concrete elements are calculated in a similar way, the difference being only in some input parameters.

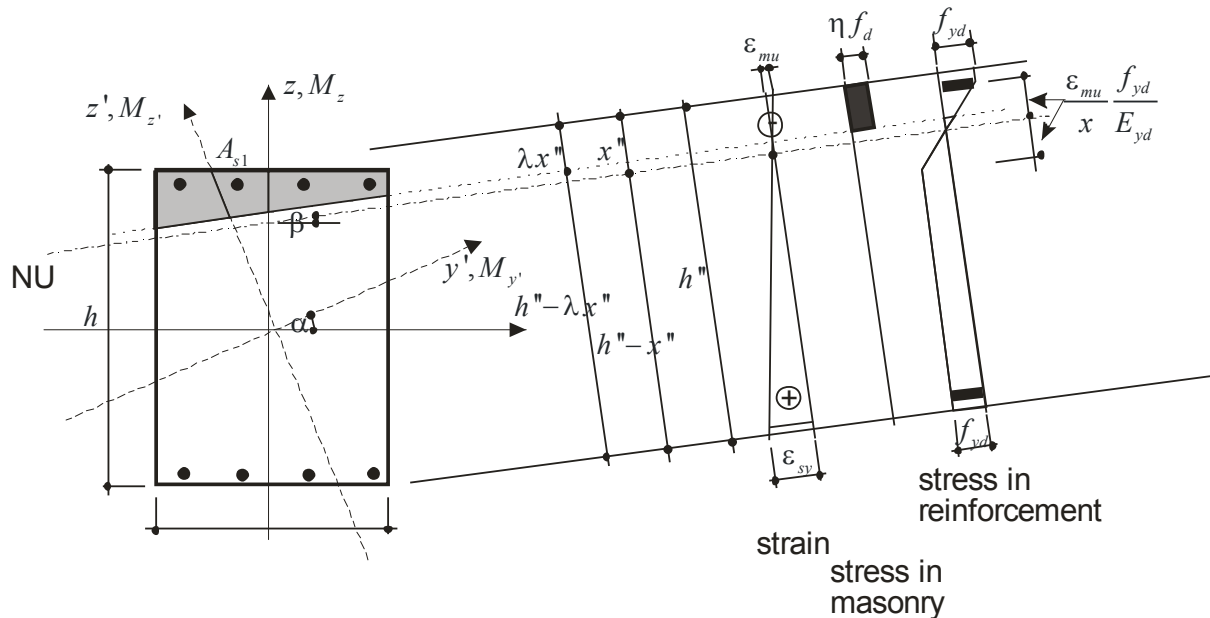


Figure 97: AmQuake beam subject to longitudinal and lateral loading

AmQuake checks design vs. ultimate loading at three locations: The bottom/left, top/right and middle cross sections. A typical cross section is depicted in Figure 97. The figure demonstrates:

- Applied/resistant load forces and moments.
- Local coordinate system x, y, z . The axis x coincides with the beam's centre axis.
- Rotated coordinate system x', y', z' . The axis x' equals to x and the axis y', z' are α angle rotated y, z so that:

$$\alpha = \text{atan}\left(\frac{M_z}{M_y}\right)$$

$$M_{y'} = \cos(\alpha)M_y + \sin(\alpha)M_z$$

$$M_{z'} = -\sin(\alpha)M_y + \cos(\alpha)M_z$$
(1.27)

Note that (1.27) yields $M_{z'} = 0$, i.e. only $M_{y'}$ needs to be checked. This is why this coordinate system is used for assessments of ultimate loading.

- Coordinate system x'', y'', z'' . The axis x'' equals to x , the axis y'' corresponds to the cross section's neutral axis (NU), i.e. the place with longitudinal strain $\varepsilon_x = 0$. This coordinate system is used to calculate strains and stresses.
- Development of the strain ε_x within the cross section. As indicated, its value depends on distance from NU, (see the point above).
- Development of stress σ_x in masonry/concrete. Tensile stress is neglected. Compression stress is assumed constant over the whole compressed area reduced by λ factor.
- η ... Default value = 0,8, For unreinforced masonry in CR6(2013) = 0,85 (0,75 for Porotherm)
- Development of stress σ_x in reinforcement. It depends on ε_x , reinforcement Young modulus E_{yd} and ultimate reinforcement strength f_{yd} . Elastic-plastic or elastic-plastic model with hardening can be used.

The procedure of checking design vs. ultimate forces/moments can be summarized in the following steps:

1. The first thing to do is to calculate the design materials' parameters from the corresponding characteristic values. It cannot be done beforehand, because it depends on actual normal force in the beam.
2. The next step is to calculate total forces and moments, for which the cross section is to be checked. Typically, this step involves addition of moments due slenderness and eccentricity, (see Eqn. 6.5 in [13]). For the top and bottom cross sections the additional eccentricity is taken as $0.05b$. For middle cross section a more elaborate expression is used. It accounts for the beam's buckling, (see Eqn. 6.8 in [13]). It also involves reduction of the loading moments due to elastic bending strength of the beam, (see Eqn. 6.15 in [13]). This reduction is, however, applicable only for out-off plane bending action.
3. In the next step, the geometry and loading of the beam is transformed to x', y', z' coordinate system. As already mentioned, in this coordinate system we have

$M_{z'} = 0$, so that for bending the cross section is checked only with respect to normal force N and moment $M_{y'}$. This step is described in more details later in this section.

4. Further, shear strength is verified. For masonry it is simply calculated as $A_c f_{vd} + A_s f_{yvd}$, where A_c is area of compressed part of the cross section, A_s is area of reinforcement and f_{vd}, f_{yvd} is respectively shear strength of base material, i.e. masonry and shear strength of reinforcement. Note that A_c is calculated based on assumption of linear elastic behaviour. For concrete and/or reinforced concrete beams this calculation is done according to Section 3.3.1 in [13], (see also Section 5.3.2 of this document).
5. The next step is to verify flexural and shear drifts of the cross section. They are defined in (1.3), (1.4) and the accompanying text and tables in Section 3.3.2. If the drifts limits are violated, then the corresponding forces/moments are set to zero. The cross-sectional behaviour is assumed to be elastic-perfectly plastic up to the point when the flexural drift limit ϕ_{lim} is reached. For masonry elements the flexural drift limit is directly specified in the program (see Figure 18). In the case of reinforced concrete elements the flexural drift limit is calculated according to EN 1998-3, A.3.2.2 as:

$$\phi_{lim} = \frac{1}{\gamma_{el}} 0.016 \cdot (0.3^v) \left[\frac{\max(0.01; \omega')}{\max(0.01; \omega)} \right]^{0.225} \left(\frac{L_v}{h} \right)^{0.35} 25^{\left(\alpha \rho_{sx} \frac{f_{yw}}{f_c} \right)} (1.25^{100 \rho_d})$$

$$\phi_{lim} = \phi_{lim}^{RC} \left[\frac{\max(0.01; \omega')}{\max(0.01; \omega)} \right]^{0.225} \left(\frac{L_v}{h} \right)^{0.35} 25^{\left(\alpha \rho_{sx} \frac{f_{yw}}{f_c} \right)} (1.25^{100 \rho_d})$$

where:

$$\phi_{lim}^{RC} = \frac{1}{\gamma_{el}} 0.016 = \frac{1}{1.5} 0.016 = 0.011, \text{ and is specified as a constant in Figure 17.}$$

$$\omega = \frac{A_s^t}{b h}, \text{ ratio of longitudinal tensile reinforcement}$$

$$\omega' = \frac{A_s^c}{b h}, \text{ ratio of longitudinal compressive reinforcement}$$

$$L_v = \frac{M}{V}, \text{ ratio of moment/shear at the end section}$$

h is the depth of the cross-section

f_c is the mean compressive strength

f_{yw} is the mean yield strength of the stirrups

$$\rho_{sx} = \frac{A_{sx}}{b_w s_h} \text{ is the ratio of transverse steel parallel to the direction of the loading,}$$

s_h is the spacing of stirrups; b_w is the effective width of the section, i.e. the distance of the longitudinal tensile reinforcement from the opposite surface.

ρ_d is the ratio of steel diagonal reinforcement, in **AmQuake** this is considered to be always 0.0.

$\alpha = \left(1 - \frac{s_h}{2b_0}\right) \left(1 - \frac{s_h}{2h_0}\right) \left(1 - \frac{\sum b_i^2}{6h_0b_0}\right)$, where b_0 , h_0 are the dimensions of the confined core to the centreline of the hoop and b_i is the centre-line spacing of the longitudinal bars that are laterally constrained by a stirrup corner or a cross-tie.

6. Check ultimate moments with respect to Eqn. (6.24) in Section 6.6.2 in [12]. If needed, reduce the moments.
7. The final step of this checking procedure is to transform all forces and moments back to the element's local coordinate system and re-establish all forces and moments equilibriums for the beam.

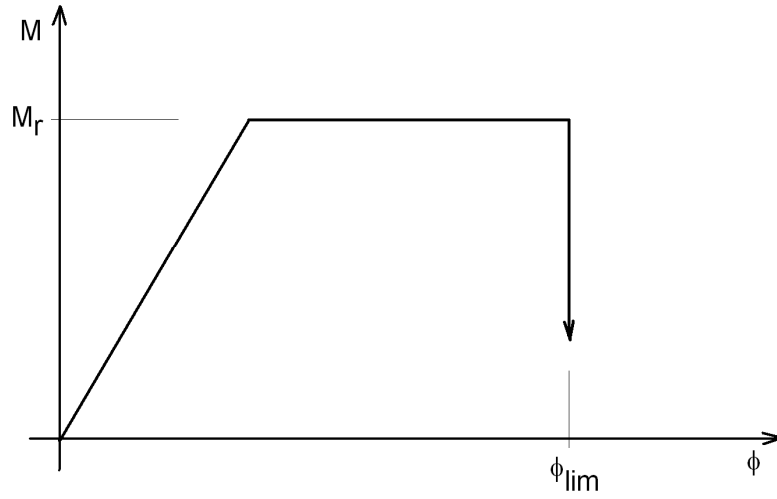


Figure 98: Idealized bi-linear behaviour of the cross-section in bending.

M-N diagram of a cross section

Step 3 of the above procedure refers to the beam's M-N assessment, i.e. checking its bending resistance. This is done in **AmQuake** using M-N diagram. Note that for each angle α a new M-N diagram is needed.

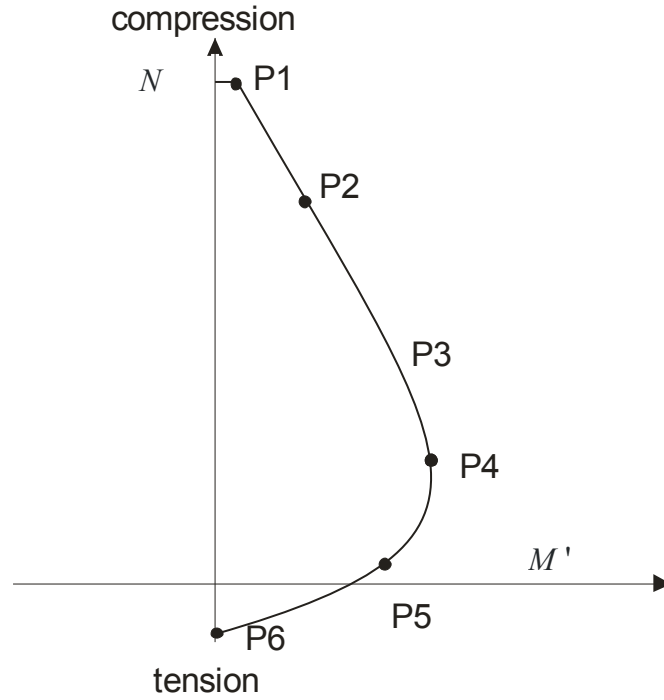


Figure 99 M-N diagram of a cross section.

Six pairs of ultimate normal force N and bending moment M' are calculated. These points are based on maximum strains at the most distant points of the cross section with respect to its neutral axis.

Point 1 is calculated for the case of uniform (compressive) beam's deformation $\varepsilon_{mu} = \varepsilon_{yd} = -0.002$ (i.e. exact value depends on the selected material), see Section 6.6.1 in [12].

Point 6 is similar to Point 1, but it assumes uniform tensile condition $\varepsilon_{mu} = \varepsilon_{yd} = \frac{f_{yd}}{E_{yd}}$, (i.e. yield strain of the reinforcement).

Points 2 assumes maximum compression $\varepsilon_{mu} = -0.0035$ (i.e. exact value depends on the selected material) and maximum tension $\varepsilon_{sy} = 0$.

Points 3 assumes maximum compression $\varepsilon_{mu} = -0.0035$ (i.e. exact value depends on the selected material) and maximum tension $\varepsilon_{yd} = \frac{f_{yd}}{E_{yd}}$.

Points 4 assumes maximum compression $\varepsilon_{mu} = -0.0035/3$. (i.e. exact value depends on the selected material) and maximum tension $\varepsilon_{yd} = f_{yd,max} = 0.01$.

Points 5 assumes maximum compression $\varepsilon_{mu} = -0.0035$ (i.e. exact value depends on the selected material) and maximum tension $\varepsilon_{yd} = f_{yd,max} = 0.01$.

The actual calculation of each of the above point is carried out iteratively. The iterating is needed, because we don't know position of y'', z'' coordinate system, (i.e. the rotation angle β) that ensures $M_{z''} = 0$.

The computed six M-N points are then fitted by a curve. For a given normal force N the curve yields ultimate moment $M_{y'}$, which is compared to the required $M_{y'}$. If the required moment is smaller than the ultimate moment, then no moment reduction is needed. Otherwise, the required model is reduced to the ultimate moment.

The whole M-N calculations can be carried out in each step and/or iteration of the solution from scratch, or (by default) **AmQuake** tries to optimize it. Each beam caches its calculated M-N diagrams (for a given angle α). Then, if it finds (for the current α) two cached M-N diagrams that are sufficiently close (with respect to α), the current M-N diagram is computed by interpolation. Otherwise, a new M-N diagram is computed and stored for eventual later use. This optimisation can speed up **AmQuake** execution by a factor of 10 or even more.

Evaluation of the damage factor CF

$$CF = 1 - \min \left[\frac{\max(\Delta F_j^i, \Delta F_{j+6}^i)}{\max(K_{jk}^i \Delta u_k^i, K_{j+6k}^i \Delta u_k^i)} \right], j = 1..6 \quad (1.28)$$

where: i is load step id, j is degree of freedom (dof) id of the left node of the beam, $j+6$ is dof id of the right node of the beam, ΔF_j^i is increment of element force at step i dof j , K_{jk}^i is element j,k of beam's stiffness matrix at step i . The outwards min() is carried out through dofs 1..6, i.e. all dofs at a node. Note that ΔF_j^i includes reduction of the forces due to material nonlinearity and damage. The factor CF is thus defined as 1 minus ratio of beam's forces increment including and excluding material nonlinearity within the current step. The most severely damaged degree of freedom is considered. It indicates, how much the element response deviate from the elastic one. It ranges from 0 to 1 for fully undamaged to fully damaged conditions.

The flags in the "Component damages table", columns "Failed dofs" mark dofs, at which the beam hits failure criterion and thus the originally calculated value based of linear elasticity had to be reduced. The dofs' failure are marked as follows:

- Nx - normal forces,
- Vy, Vz - shear force in Y,Z direction,
- Kx - moment in torque,
- My, Mz - bending moments in Y,Z direction,
- SD - total shear capacity exhausted, full shear failure,
- FD - total flexural capacity exhausted, full flexural failure,
- MV - both moment and shear ultimate forces are reached, plastic behaviour in flexure and shear,
- M - moment capacity reached, plastic behaviour in bending/flexure,
- V - shear capacity reached, plastic behaviour in shear

The SD and FD status flags refer to so called shear and flexural limits, see Eqns. (1.3-4).

6 CONCLUSIONS

The objective of the software is to provide a seismic assessment of masonry buildings using the pushover analysis and the equivalent frame method.

The program installation is described in Section 2.

The description of the graphical user interface is in Section 2.3.

Theoretical basis for the pushover analysis, equivalent frame method is described in Section 5.

Various examples can be found in AmQuake installation directory. It is recommended to try and study these example problems before using AmQuake for your own building projects.

7 LITERATURE

- [1] PhD Thesis of Paolo Morandi, "New Proposals for Simplified Seismic Design of Masonry Buildings, November 2006, University of Pavia and ROSE School
- [2] Eurocode 6 (CR6 Romania), Design of Masonry Structures,
- [3] Eurocode 8 (P100 Romania), Design of Structures for Earthquake resistance,
- [4] J. Sejnoha, "Pružnost, pevnost, plasticita !" Prague: CTU, 1979.
- [5] K. autoru, "Zaklady programování MKP", Prague: Dum techniky CSVTS, 1981. Z. Bittnar and J. Sejnoha, "Numerical Methods in Structural Mechanics": Thomas Telford, 1996
- [6] K.J. Bathe, "Finite Element Procedures in Engineering Analysis", Englewood Cliffs, New Jersey 07632: Prentice Hall, Inc., 1982
- [7] V. Cervenka, J. Cervenka, and L. Jendele, "Atena Program Documentation, Part 1 Theory", Prague: Cervenka Consl., 2000-2007
- [8] L. Jendele and J. Cervenka, "On the Solution of Multi-Point Constraints - Application to FE Analysis of Reinforced Concrete Structures", Computer and Structures, 2006
- [9] Z. Bittnar, J. Sejnoha, "Numerical Methods in Structural Mechanics", ISBN 0-7844-0170-5, ASCE Press, 1996
- [10] Eurocode 6: Design of Masonry Structures-Part 1-1" Common rules for reinforced and unreinforced structures.
- [11] Eurocode 2: Design of Reinforced Concrete
- [12] J. Cervenka, AmQuake Validation and Testing, Cervenka Consulting, 2010

8 APPENDIX A

This is the contents of the national coefficient table for the various national standards. This table is loaded by the program everytime a new data file is created.

parametr	Austria	Belgium	Bulgaria	Croatia	Czech republic	France	German	Hungary	Romani Eurocod	Romani CR6+ P100	Slovak republic	Slovenia	description
gamma_C_RC_EN2_1_1	1,5	1,5	1,5	1,5	1,5	1,5	1,5	1,5	1,5	1,5	1,5	1,5	Partial safety factor for concrete for static analysis
gamma_S_RC_EN2_1_1	1,15	1,15	1,15	1,15	1,15	1,15	1,15	1,15	1,15	1,15	1,15	1,15	Partial safety factor for steel for static analysis
fck_EN2_1_1_C12	12	12	12	12	12	12	12	12	12	12	12	12	Characteristic concrete compressive strength, C12
ec_EN2_1_1_C12	27000	27000	27000	27000	27000	27000	27000	27000	27000	27000	27000	27000	Young modulus of concrete C12, mean
gc_EN2_1_1_C12	11250	11250	11250	11250	11250	11250	11250	11250	11250	11250	11250	11250	Shear modulus of concrete C12, mean
fcvk_EN2_1_1_C12	0,27	0,27	0,27	0,27	0,27	0,27	0,27	0,27	0,27	0,27	0,27	0,27	Shear character. strength of concrete C12
fck_EN2_1_1_C16	16	16	16	16	16	16	16	16	16	16	16	16	Characteristic concrete compressive strength, C16
ec_EN2_1_1_C16	29000	29000	29000	29000	29000	29000	29000	29000	29000	29000	29000	29000	Young modulus of concrete C16, mean
gc_EN2_1_1_C16	12083	12083	12083	12083	12083	12083	12083	12083	12083	12083	12083	12083	Shear modulus of concrete C16, mean
fcvk_EN2_1_1_C16	0,33	0,33	0,33	0,33	0,33	0,33	0,33	0,33	0,33	0,33	0,33	0,33	Shear character. strength of concrete C16
fck_EN2_1_1_C20	20	20	20	20	20	20	20	20	20	20	20	20	Characteristic concrete compressive strength, Cxx
ec_EN2_1_1_C20	30000	30000	30000	30000	30000	30000	30000	30000	30000	30000	30000	30000	Young modulus of concrete Cxx, mean
gc_EN2_1_1_C20	12500	12500	12500	12500	12500	12500	12500	12500	12500	12500	12500	12500	Shear modulus of concrete Cxx, mean
fcvk_EN2_1_1_C20	0,39	0,39	0,39	0,39	0,39	0,39	0,39	0,39	0,39	0,39	0,39	0,39	Shear character. strength of concrete Cxx

fck_EN2_1_1_C25	25	25	25	25	25	25	25	25	25	25	25	25	Characteristic concrete compressive strength, Cxx
ec_EN2_1_1_C25	31000	31000	31000	31000	31000	31000	31000	31000	31000	31000	31000	31000	Young modulus of concrete Cxx, mean
gc_EN2_1_1_C25	12917	12917	12917	12917	12917	12917	12917	12917	12917	12917	12917	12917	Shear modulus of concrete Cxx, mean
fcvk_EN2_1_1_C25	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	Shear character. strength of concrete Cxx
fck_EN2_1_1_C30	30	30	30	30	30	30	30	30	30	30	30	30	Characteristic concrete compressive strength, Cxx
ec_EN2_1_1_C30	33000	33000	33000	33000	33000	33000	33000	33000	33000	33000	33000	33000	Young modulus of concrete Cxx, mean
gc_EN2_1_1_C30	13750	13750	13750	13750	13750	13750	13750	13750	13750	13750	13750	13750	Shear modulus of concrete Cxx, mean
fcvk_EN2_1_1_C30	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	Shear character. strength of concrete Cxx
fck_EN2_1_1_C35	35	35	35	35	35	35	35	35	35	35	35	35	Characteristic concrete compressive strength, Cxx
ec_EN2_1_1_C35	34000	34000	34000	34000	34000	34000	34000	34000	34000	34000	34000	34000	Young modulus of concrete Cxx, mean
gc_EN2_1_1_C35	14167	14167	14167	14167	14167	14167	14167	14167	14167	14167	14167	14167	Shear modulus of concrete Cxx, mean
fcvk_EN2_1_1_C35	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	Shear character. strength of concrete Cxx
fck_EN2_1_1_C40	40	40	40	40	40	40	40	40	40	40	40	40	Characteristic concrete compressive strength, Cxx
ec_EN2_1_1_C40	35000	35000	35000	35000	35000	35000	35000	35000	35000	35000	35000	35000	Young modulus of concrete Cxx, mean
gc_EN2_1_1_C40	14583	14583	14583	14583	14583	14583	14583	14583	14583	14583	14583	14583	Shear modulus of concrete Cxx, mean
fcvk_EN2_1_1_C40	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	Shear character. strength of concrete Cxx
fck_EN2_1_1_C45	45	45	45	45	45	45	45	45	45	45	45	45	Characteristic concrete compressive strength, Cxx
ec_EN2_1_1_C45	36000	36000	36000	36000	36000	36000	36000	36000	36000	36000	36000	36000	Young modulus of concrete Cxx, mean
gc_EN2_1_1_C45	15000	15000	15000	15000	15000	15000	15000	15000	15000	15000	15000	15000	Shear modulus of concrete Cxx, mean
fcvk_EN2_1_1_C45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	Shear character. strength of concrete Cxx
fck_EN2_1_1_C50	50	50	50	50	50	50	50	50	50	50	50	50	Characteristic concrete compressive strength, Cxx

ec_EN2_1_1_C50	37000	37000	37000	37000	37000	37000	37000	37000	37000	37000	37000	37000	Young modulus of concrete Cxx, mean
gc_EN2_1_1_C50	15417	15417	15417	15417	15417	15417	15417	15417	15417	15417	15417	15417	Shear modulus of concrete Cxx, mean
fcvk_EN2_1_1_C50	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	Shear character. strength of concrete Cxx
fck_EN2_1_1_C55	55	55	55	55	55	55	55	55	55	55	55	55	Characteristic concrete compressive strength, Cxx
ec_EN2_1_1_C55	38000	38000	38000	38000	38000	38000	38000	38000	38000	38000	38000	38000	Young modulus of concrete Cxx, mean
gc_EN2_1_1_C55	15833	15833	15833	15833	15833	15833	15833	15833	15833	15833	15833	15833	Shear modulus of concrete Cxx, mean
fcvk_EN2_1_1_C55	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	Shear character. strength of concrete Cxx
fck_EN2_1_1_C60	60	60	60	60	60	60	60	60	60	60	60	60	Characteristic concrete compressive strength, Cxx
ec_EN2_1_1_C60	39000	39000	39000	39000	39000	39000	39000	39000	39000	39000	39000	39000	Young modulus of concrete Cxx, mean
gc_EN2_1_1_C60	16250	16250	16250	16250	16250	16250	16250	16250	16250	16250	16250	16250	Shear modulus of concrete Cxx, mean
fcvk_EN2_1_1_C60	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	Shear character. strength of concrete Cxx
fck_EN2_1_1_C70	70	70	70	70	70	70	70	70	70	70	70	70	Characteristic concrete compressive strength, Cxx
ec_EN2_1_1_C70	41000	41000	41000	41000	41000	41000	41000	41000	41000	41000	41000	41000	Young modulus of concrete Cxx, mean
gc_EN2_1_1_C70	17083	17083	17083	17083	17083	17083	17083	17083	17083	17083	17083	17083	Shear modulus of concrete Cxx, mean
fcvk_EN2_1_1_C70	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	Shear character. strength of concrete Cxx
fck_EN2_1_1_C80	80	80	80	80	80	80	80	80	80	80	80	80	Characteristic concrete compressive strength, Cxx
ec_EN2_1_1_C80	42000	42000	42000	42000	42000	42000	42000	42000	42000	42000	42000	42000	Young modulus of concrete Cxx, mean
gc_EN2_1_1_C80	17500	17500	17500	17500	17500	17500	17500	17500	17500	17500	17500	17500	Shear modulus of concrete Cxx, mean
fcvk_EN2_1_1_C80	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	Shear character. strength of concrete Cxx
fck_EN2_1_1_C90	90	90	90	90	90	90	90	90	90	90	90	90	Characteristic concrete compressive strength, Cxx
ec_EN2_1_1_C90	44000	44000	44000	44000	44000	44000	44000	44000	44000	44000	44000	44000	Young modulus of concrete Cxx, mean

[illegible]

eps_c3_EN2_1_1_C50	-0,00175	-0,00175	-0,00175	-0,00175	-0,00175	-0,00175	-0,00175	-0,00175	-0,00175	-0,00175	-0,00175	-0,00175	Eps_c3 of concrete Cxx
eps_c3_EN2_1_1_C55	-0,0018	-0,0018	-0,0018	-0,0018	-0,0018	-0,0018	-0,0018	-0,0018	-0,0018	-0,0018	-0,0018	-0,0018	Eps_c3 of concrete Cxx
eps_c3_EN2_1_1_C60	-0,0019	-0,0019	-0,0019	-0,0019	-0,0019	-0,0019	-0,0019	-0,0019	-0,0019	-0,0019	-0,0019	-0,0019	Eps_c3 of concrete Cxx
eps_c3_EN2_1_1_C70	-0,002	-0,002	-0,002	-0,002	-0,002	-0,002	-0,002	-0,002	-0,002	-0,002	-0,002	-0,002	Eps_c3 of concrete Cxx
eps_c3_EN2_1_1_C80	-0,0022	-0,0022	-0,0022	-0,0022	-0,0022	-0,0022	-0,0022	-0,0022	-0,0022	-0,0022	-0,0022	-0,0022	Eps_c3 of concrete Cxx
eps_c3_EN2_1_1_C90	-0,0023	-0,0023	-0,0023	-0,0023	-0,0023	-0,0023	-0,0023	-0,0023	-0,0023	-0,0023	-0,0023	-0,0023	Eps_c3 of concrete Cxx
fcvkfcvm	0,833	0,833	0,833	0,833	0,833	0,833	0,833	0,833	0,833	0,833	0,833	0,833	Ratio of char to mean shear strength
fmkfm	0,833	0,833	0,833	0,833	0,833	0,833	0,833	0,833	0,833	0,833	0,833	0,833	Ratio of char. to mean yield strength of masonry
cover	0,03	0,03	0,03	0,03	0,03	0,03	0,03	0,03	0,03	0,03	0,03	0,03	Default cover for concrete
gamma_M_EN6_1_1	2,2	2,2	2,2	2,2	2	2,2	2,2	2,2	2,2	2,2	2,2	2,2	Partial safety factor for masonry for static analysis, see EN 1996-1-1 sec. 2.4.1 For Class 3 - prescribed mortar
k_E	1000	1000	1000	1000	1000	1000	1000	1000	1000	1000	1000	1000	EN 1996-1-1, 3.7.2, Used to calculate masonry wall elastic modulus E
k_G	0,4	0,4	0,4	0,4	0,4	0,4	0,4	0,4	0,4	0,4	0,4	0,4	EN 1996-1-1, 3.7.3, Used to calculate masonry wall shear modulus G
k_Table_3_3_EN6_1_1_e1_1	0,6	0,55	0,55	0,55	0,55	0,55	0,55	0,55	0,55	0,4	0,55	0,55	EN 1996-1-1, Table 3.3, element (1,1)
k_Table_3_3_EN6_1_1_e2_1	0,55	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,45	0,4	0,45	0,45	EN 1996-1-1, Table 3.3, element (2,1)
k_Table_3_3_EN6_1_1_e3_1	0,5	0,35	0,35	0,35	0,35	0,35	0,35	0,35	0,35	0,4	0,35	0,35	EN 1996-1-1, Table 3.3, element (3,1)
k_Table_3_3_EN6_1_1_e4_1	0,5	0,35	0,35	0,35	0,35	0,35	0,35	0,35	0,35	0,35	0,35	0,35	EN 1996-1-1, Table 3.3, element (4,1)
k_Table_3_3_EN6_1_1_e1_2	0,9	0,75	0,75	0,75	0,75	0,75	0,75	0,75	0,75	0,75	0,75	0,75	EN 1996-1-1, Table 3.3, element (1,2)
k_Table_3_3_EN6_1_1_e2_2	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	EN 1996-1-1, Table 3.3, element (2,2)
k_Table_3_3_EN6_1_1_e3_2	0,5	0,5	0,5	0,5	0,5	0,5	0,5	0,5	0,5	0,5	0,5	0,5	EN 1996-1-1, Table 3.3, element (3,2)
k_Table_3_3_EN6_1_1_e4_2	0,5	0,35	0,35	0,35	0,35	0,35	0,35	0,35	0,35	0,35	0,35	0,35	EN 1996-1-1, Table 3.3, element (4,2)
k_Table_3_3_EN6_1_1_e1_3	0,35	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,25	0,3	0,3	EN 1996-1-1, Table 3.3, element (1,3)
k_Table_3_3_EN6_1_1_e2_3	0,3	0,25	0,25	0,25	0,25	0,25	0,25	0,25	0,25	0,25	0,25	0,25	EN 1996-1-1, Table 3.3, element (2,3)

k_Table 3 3 EN6_1_1_e3_3	0,25	0,2	0,2	0,2	0,2	0,2	0,2	0,2	0,2	0,25	0,2	0,2	EN 1996-1-1, Table 3.3, element (3,3)
k_Table 3 3 EN6_1_1_e4_3	0,25	0,2	0,2	0,2	0,2	0,2	0,2	0,2	0,2	0,25	0,2	0,2	EN 1996-1-1, Table 3.3, element (4,3)
r_Eq_3_6_EN6_1_1_tg	0,5	0,5	0,5	0,5	0,5	0,5	0,5	0,5	0,5	0,5	0,5	0,5	EN 1996-1-1, Coeff. before f_vko in equation 3.6 for tongue and groove units
r_Eq_3_6_EN6_1_1_def	1	1	1	1	1	1	1	1	1	1	1	1	EN 1996-1-1, Coeff. before f_vko in equation 3.6 - default case
f_vlt_EN6_1_1_tg	0,045	0,045	0,045	0,045	0,045	0,045	0,045	0,045	0,045	0,0306	0,045	0,045	EN 1996-1-1, f_vlt for tongue and groove units
f_vlt_EN6_1_1_def	0,065	0,065	0,065	0,065	0,065	0,065	0,065	0,065	0,065	0,0306	0,065	0,065	EN 1996-1-1, f_vlt for all other cases
f_vk0_EN6_1_1_Table_3_4_e1_1	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	EN 1996-1-1, f_vk0 from Table 3.4, element(1,1)
f_vk0_EN6_1_1_Table_3_4_e2_1	0,2	0,2	0,2	0,2	0,2	0,2	0,2	0,2	0,2	0,2	0,2	0,2	EN 1996-1-1, f_vk0 from Table 3.4, element(2,1)
f_vk0_EN6_1_1_Table_3_4_e1_2	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	EN 1996-1-1, f_vk0 from Table 3.4, element(1,2)
f_vk0_EN6_1_1_Table_3_4_e1_3	0,15	0,15	0,15	0,15	0,15	0,15	0,15	0,15	0,15	0,15	0,15	0,15	EN 1996-1-1, f_vk0 from Table 3.4, element(1,3)
f_xk1_EN6_1_1_e1_2	0,1	0,1	0,1	0,1	0,1	0,1	0,1	0,1	0,24	0,24	0,1	0,1	EN 1996-1-1. f_xk1, element(1,2)
f_xk1_EN6_1_1_e1_3	0,15	0,15	0,15	0,15	0,15	0,15	0,15	0,15	0,15	0,15	0,15	0,15	EN 1996-1-1. f_xk1, element(1,3)
f_xk1_EN6_1_1_e1_4	0,1	0,1	0,1	0,1	0,1	0,1	0,1	0,1	0,1	0,1	0,1	0,1	EN 1996-1-1. f_xk1, element(1,4)
f_xk2_EN6_1_1_e1_2	0,4	0,4	0,4	0,4	0,4	0,4	0,4	0,4	0,48	0,48	0,4	0,4	EN 1996-1-1. f_xk2, element(1,2)
f_xk2_EN6_1_1_e1_3	0,15	0,15	0,15	0,15	0,15	0,15	0,15	0,15	0,15	0,15	0,15	0,15	EN 1996-1-1. f_xk2, element(1,3)
f_xk2_EN6_1_1_e1_4	0,1	0,1	0,1	0,1	0,1	0,1	0,1	0,1	0,1	0,1	0,1	0,1	EN 1996-1-1. f_xk2, element(1,4)
gamma_I_EN8_1_Sec_4_2_5_I_zone0	0,8	0,8	0,8	0,8	0,8	0,8	0,8	0,8	0,8	1,4	0,8	0,8	EN 1998-1, Gamma_I for class I, Sec. 4.2.5 zone 0
gamma_I_EN8_1_Sec_4_2_5_I_zone1	0,8	0,8	0,8	0,8	0,8	0,8	0,8	0,8	0,8	1,4	0,8	0,8	EN 1998-1, Gamma_I for class I, Sec. 4.2.5 zone 1
gamma_I_EN8_1_Sec_4_2_5_I_zone2	0,8	0,8	0,8	0,8	0,8	0,8	0,8	0,8	0,8	1,4	0,8	0,8	EN 1998-1, Gamma_I for class I, Sec. 4.2.5 zone 2
gamma_I_EN8_1_Sec_4_2_5_I_zone3	0,8	0,8	0,8	0,8	0,8	0,8	0,8	0,8	0,8	1,4	0,8	0,8	EN 1998-1, Gamma_I for class I, Sec. 4.2.5 zone 3

gamma_I_EN8_1_Sec_4_2_5_I_zone4	0,8	0,8	0,8	0,8	0,8	0,8	0,8	0,8	0,8	1,4	0,8	0,8	EN 1998-1, Gamma_I for class I, Sec. 4.2.5 zone 4
gamma_I_EN8_1_Sec_4_2_5_II_zone0	1	1	1	1	1	1	1	1	1	1,2	1	1	EN 1998-1, Gamma_I for class II, Sec. 4.2.5 zone 0
gamma_I_EN8_1_Sec_4_2_5_II_zone1	1	1	1	1	1	1	1	1	1	1,2	1	1	EN 1998-1, Gamma_I for class II, Sec. 4.2.5 zone 1
gamma_I_EN8_1_Sec_4_2_5_II_zone2	1	1	1	1	1	1	1	1	1	1,2	1	1	EN 1998-1, Gamma_I for class II, Sec. 4.2.5 zone 2
gamma_I_EN8_1_Sec_4_2_5_II_zone3	1	1	1	1	1	1	1	1	1	1,2	1	1	EN 1998-1, Gamma_I for class II, Sec. 4.2.5 zone 3
gamma_I_EN8_1_Sec_4_2_5_II_zone4	1	1	1	1	1	1	1	1	1	1,2	1	1	EN 1998-1, Gamma_I for class II, Sec. 4.2.5 zone 4
gamma_I_EN8_1_Sec_4_2_5_III_zone0	1	1,2	1,2	1,2	1,2	1,2	1,2	1,2	1,2	1	1,2	1,2	EN 1998-1, Gamma_I for class III, Sec. 4.2.5 zone 0
gamma_I_EN8_1_Sec_4_2_5_III_zone1	1	1,2	1,2	1,2	1,2	1,2	1,2	1,2	1,2	1	1,2	1,2	EN 1998-1, Gamma_I for class III, Sec. 4.2.5 zone 1
gamma_I_EN8_1_Sec_4_2_5_III_zone2	1,1	1,2	1,2	1,2	1,2	1,2	1,2	1,2	1,2	1	1,2	1,2	EN 1998-1, Gamma_I for class III, Sec. 4.2.5 zone 2
gamma_I_EN8_1_Sec_4_2_5_III_zone3	1,4	1,2	1,2	1,2	1,2	1,2	1,2	1,2	1,2	1	1,2	1,2	EN 1998-1, Gamma_I for class III, Sec. 4.2.5 zone 3
gamma_I_EN8_1_Sec_4_2_5_III_zone4	1,4	1,2	1,2	1,2	1,2	1,2	1,2	1,2	1,2	1	1,2	1,2	EN 1998-1, Gamma_I for class III, Sec. 4.2.5 zone 4
gamma_I_EN8_1_Sec_4_2_5_IV_zone0	1	1,4	1,4	1,4	1,4	1,4	1,4	1,4	1,4	0,8	1,4	1,4	EN 1998-1, Gamma_I for class IV, Sec. 4.2.5 zone 0
gamma_I_EN8_1_Sec_4_2_5_IV_zone1	1	1,4	1,4	1,4	1,4	1,4	1,4	1,4	1,4	0,8	1,4	1,4	EN 1998-1, Gamma_I for class IV, Sec. 4.2.5 zone 1
gamma_I_EN8_1_Sec_4_2_5_IV_zone2	1,2	1,4	1,4	1,4	1,4	1,4	1,4	1,4	1,4	0,8	1,4	1,4	EN 1998-1, Gamma_I for class IV, Sec. 4.2.5 zone 2
gamma_I_EN8_1_Sec_4_2_5_IV_zone3	1,4	1,4	1,4	1,4	1,4	1,4	1,4	1,4	1,4	0,8	1,4	1,4	EN 1998-1, Gamma_I for class IV, Sec. 4.2.5 zone 3
gamma_I_EN8_1_Sec_4_2_5_IV_zone4	1,4	1,4	1,4	1,4	1,4	1,4	1,4	1,4	1,4	0,8	1,4	1,4	EN 1998-1, Gamma_I for class IV, Sec. 4.2.5 zone 4
crackedWallCoeff_EN8_Sec_4_3_1	0,5	0,5	0,5	0,5	0,5	0,5	0,5	0,5	0,5	0,5	0,5	0,5	Cracked wall stiffness coefficient (EN 1998-1, 4.3.1(7))

phi_fvk	0,4	0,4	0,4	0,4	0,4	0,4	0,4	0,4	0,4	0,4	0,4	0,4	phi_fvk for calculation of fvk for masonry
phi_fvlt	0	0	0	0	0	0	0	0	0	0,126	0	0	phi_fvlt for calculation of fvlt for masonry
sum_of_seismic_zone	31	31	1	1	1	1	1	1	1	1	1	1	0=+1 1=+2 2=+4 3=+8 4=+16
sum_of_spectrum_type_soil1	1	2	3	3	3	3	3	3	3	3	3	1	1 - type 1, 3 - type 1+2
sum_of_spectrum_type_soil2	1	2	3	3	3	3	3	3	1	1	3	1	1 - type 1, 3 - type 1+2
sum_of_spectrum_type_soil3	1	2	3	3	3	3	3	3	1	1	3	1	1 - type 1, 3 - type 1+2
sum_of_spectrum_type_soil4	1	2	3	3	3	3	3	3	0	0	3	1	1 - type 1, 3 - type 1+2
sum_of_spectrum_type_soil5	1	2	3	3	3	3	3	3	0	0	3	1	1 - type 1, 3 - type 1+2
sum_of_soil_type	5	5	5	5	5	5	5	5	3	3	5	5	
soil_type_1	A	A	A	A	A	A	A	A	Z1-Tc 0,7	Z1-Tc 0,7	A	A	
soil_type_2	B	B	B	B	B	B	B	B	Z2-Tc 1,0	Z2-Tc 1,0	B	B	
soil_type_3	C	C	C	C	C	C	C	C	Z3-Tc 1,6	Z3-Tc 1,6	C	C	
soil_type_4	D	D	D	D	D	D	D	D	-	-	D	D	
soil_type_5	E	E	E	E	E	E	E	E	-	-	E	E	
dynamic_amplification_beta0_spectrum1	2,5	2,5	2,5	2,5	2,5	2,5	2,5	2,5	2,75	2,75	2,5	2,5	
dynamic_amplification_beta0_spectrum2	2,5	2,5	2,5	2,5	2,5	2,5	2,5	2,5	3	3	2,5	2,5	
table_3_2_EN8_1_e1_1	1	1	1	1	1	1	1	1	1	1	1	1	Table 3.2 EN 1998-1, element(1,1)
table_3_2_EN8_1_e2_1	1,2	1,2	1,3	1,2	1,2	1,2	1,2	1,2	1,2	1,2	1,1	1,2	Table 3.2 EN 1998-1, element(2,1)
table_3_2_EN8_1_e3_1	1,15	1,15	1,2	1,15	1,15	1,15	1,15	1,15	1,15	1,15	1,25	1,15	Table 3.2 EN 1998-1, element(3,1)
table_3_2_EN8_1_e4_1	1,35	1,35	1	1,35	1,35	1,35	1,35	1,35	1,35	1,35	1,5	1,35	Table 3.2 EN 1998-1, element(4,1)
table_3_2_EN8_1_e5_1	1,4	1,4	1,2	1,7	1,4	1,4	1,4	1,4	1,4	1,4	1,32	1,7	Table 3.2 EN 1998-1, element(5,1)
table_3_2_EN8_1_e1_2	0,15	0,15	0,1	0,1	0,15	0,15	0,15	0,15	0,07	0,07	0,1	0,1	Table 3.2 EN 1998-1, element(1,2)
table_3_2_EN8_1_e2_2	0,15	0,15	0,1	0,15	0,15	0,15	0,15	0,15	0,1	0,1	0,11	0,15	Table 3.2 EN 1998-1, element(2,2)

table_3_2_EN8_1_e3_2	0,2	0,2	0,1	0,2	0,2	0,2	0,2	0,2	0,16	0,16	0,125	0,2	Table 3.2 EN 1998-1, element(3,2)
table_3_2_EN8_1_e4_2	0,2	0,2	0,1	0,2	0,2	0,2	0,2	0,2	0,2	0,2	0,125	0,2	Table 3.2 EN 1998-1, element(4,2)
table_3_2_EN8_1_e5_2	0,15	0,15	0,1	0,1	0,15	0,15	0,15	0,15	0,15	0,15	0,11	0,1	Table 3.2 EN 1998-1, element(5,2)
table_3_2_EN8_1_e1_3	0,4	0,4	0,3	0,4	0,4	0,4	0,4	0,4	0,7	0,7	0,33	0,4	Table 3.2 EN 1998-1, element(1,3)
table_3_2_EN8_1_e2_3	0,5	0,5	0,4	0,5	0,5	0,5	0,5	0,5	1	1	0,64	0,5	Table 3.2 EN 1998-1, element(2,3)
table_3_2_EN8_1_e3_3	0,6	0,6	0,5	0,6	0,6	0,6	0,6	0,6	1,6	1,6	1	0,6	Table 3.2 EN 1998-1, element(3,3)
table_3_2_EN8_1_e4_3	0,8	0,8	0,6	0,8	0,8	0,8	0,8	0,8	0,8	0,8	1,25	0,8	Table 3.2 EN 1998-1, element(4,3)
table_3_2_EN8_1_e5_3	0,5	0,5	0,5	0,4	0,5	0,5	0,5	0,5	0,5	0,5	0,64	0,4	Table 3.2 EN 1998-1, element(5,3)
table_3_2_EN8_1_e1_4	2	2	2	2	2	2	2	2	3	3	1,25	2	Table 3.2 EN 1998-1, element(1,4)
table_3_2_EN8_1_e2_4	2	2	2	2	2	2	2	2	3	3	2	2	Table 3.2 EN 1998-1, element(2,4)
table_3_2_EN8_1_e3_4	2	2	2	2	2	2	2	2	2	2	3	2	Table 3.2 EN 1998-1, element(3,4)
table_3_2_EN8_1_e4_4	2	2	2	2	2	2	2	2	2	2	4	2	Table 3.2 EN 1998-1, element(4,4)
table_3_2_EN8_1_e5_4	2	2	2	2	2	2	2	2	2	2	2	2	Table 3.2 EN 1998-1, element(5,4)
table_3_3_EN8_1_e1_1	1	1	1	1	1	1	1	1	1	1	1	1	Table 3.3 EN 1998-1, element(1,1)
table_3_3_EN8_1_e2_1	1,35	1,35	1,3	1,35	1,35	1,35	1,35	1,35	1,35	1,35	1,35	1,35	Table 3.3 EN 1998-1, element(2,1)
table_3_3_EN8_1_e3_1	1,5	1,5	1,2	1,5	1,5	1,5	1,5	1,5	1,5	1,5	1,5	1,5	Table 3.3 EN 1998-1, element(3,1)
table_3_3_EN8_1_e4_1	1,8	1,8	1	1,8	1,8	1,8	1,8	1,8	1,8	1,8	1,8	1,8	Table 3.3 EN 1998-1, element(4,1)
table_3_3_EN8_1_e5_1	1,6	1,6	1,2	1,6	1,6	1,6	1,6	1,6	1,6	1,6	1,6	1,6	Table 3.3 EN 1998-1, element(5,1)
table_3_3_EN8_1_e1_2	0,05	0,05	0,1	0,05	0,05	0,05	0,05	0,05	0,07	0,07	0,05	0,05	Table 3.3 EN 1998-1, element(1,2)
table_3_3_EN8_1_e2_2	0,05	0,05	0,1	0,05	0,05	0,05	0,05	0,05	0,05	0,05	0,05	0,05	Table 3.3 EN 1998-1, element(2,2)
table_3_3_EN8_1_e3_2	0,1	0,1	0,1	0,1	0,1	0,1	0,1	0,1	0,1	0,1	0,1	0,1	Table 3.3 EN 1998-1, element(3,2)
table_3_3_EN8_1_e4_2	0,1	0,1	0,1	0,1	0,1	0,1	0,1	0,1	0,1	0,1	0,1	0,1	Table 3.3 EN 1998-1, element(4,2)
table_3_3_EN8_1_e5_2	0,05	0,05	0,1	0,05	0,05	0,05	0,05	0,05	0,05	0,05	0,05	0,05	Table 3.3 EN 1998-1, element(5,2)
table_3_3_EN8_1_e1_3	0,25	0,25	0,3	0,25	0,25	0,25	0,25	0,25	0,7	0,7	0,25	0,25	Table 3.3 EN 1998-1, element(1,3)
table_3_3_EN8_1_e2_3	0,25	0,25	0,4	0,25	0,25	0,25	0,25	0,25	0,25	0,25	0,25	0,25	Table 3.3 EN 1998-1, element(2,3)
table_3_3_EN8_1_e3_3	0,25	0,25	0,5	0,25	0,25	0,25	0,25	0,25	0,25	0,25	0,25	0,25	Table 3.3 EN 1998-1, element(3,3)
table_3_3_EN8_1_e4_3	0,3	0,3	0,6	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	Table 3.3 EN 1998-1, element(4,3)

table_3_3_EN8_1_e5_3	0,25	0,25	0,5	0,25	0,25	0,25	0,25	0,25	0,25	0,25	0,25	0,25	Table 3.3 EN 1998-1, element(5,3)
table_3_3_EN8_1_e1_4	1,2	1,2	2	1,2	1,2	1,2	1,2	1,2	3	3	1,2	1,2	Table 3.3 EN 1998-1, element(1,4)
table_3_3_EN8_1_e2_4	1,2	1,2	2	1,2	1,2	1,2	1,2	1,2	1,2	1,2	1,2	1,2	Table 3.3 EN 1998-1, element(2,4)
table_3_3_EN8_1_e3_4	1,2	1,2	2	1,2	1,2	1,2	1,2	1,2	1,2	1,2	1,2	1,2	Table 3.3 EN 1998-1, element(3,4)
table_3_3_EN8_1_e4_4	1,2	1,2	2	1,2	1,2	1,2	1,2	1,2	1,2	1,2	1,2	1,2	Table 3.3 EN 1998-1, element(4,4)
table_3_3_EN8_1_e5_4	1,2	1,2	2	1,2	1,2	1,2	1,2	1,2	1,2	1,2	1,2	1,2	Table 3.3 EN 1998-1, element(5,4)
eccentricity_of_load	5	5	5	5	5	5	5	5	5	5	5	5	% eccentricity for load
lambda_EN8_1_Sec4_3_3_2_2_a	0,85	0,85	0,85	0,85	0,85	0,85	0,85	0,85	0,85	0,85	0,85	0,85	Case (a) in Lambda determination in EN 1998-1, Sec. 4.3.3.2.2
lambda_EN8_1_Sec4_3_3_2_2_b	1	1	1	1	1	1	1	1	1	1	1	1	Case (b) in Lambda determination in EN 1998-1, Sec. 4.3.3.2.2
pd_EN8_1_Sec4_3_3_4_2_3	1,5	1,5	1,5	1,5	1,5	1,5	1,5	1,5	1,5	1,5	1,5	1,5	Capacity curve should go up to this relative value of the target displacement, EN 1998-1, 4.3.3.4.2.3
pf_EN8_1_Sec4_3_3_4_2_3	0,8	0,8	0,8	0,8	0,8	0,8	0,8	0,8	0,8	0,8	0,8	0,8	Strength of the capacity curve should not drop below this relative value of the peak strength, EN 1998-3:2005, C.3.3(2)
ground_Acceleration	1	0,9	0,9	0,9	0,9	0,9	0,9	0,9	0,9	0,9	0,9	0,9	Default ground acceleration
damping_EN8_1_Sec3_2_2_2_1	5	5	5	5	5	5	5	5	5	8	5	5	Default viscous damping EN 1998-1, Sec. 3.2.2.2 (1)
gamma_D_EN8_1_Sec2_1_4	0,584	0,584	0,584	0,584	0,584	0,584	0,584	0,584	0,584	0,584	0,584	0,584	Default value of the importance factor for damage limitation requirement
dlsReductionFactor_I	0,5	0,5	0,5	0,5	0,5	0,5	0,5	0,5	0,5	0,5	0,5	0,5	DLS reduction factor EN 1998-1, Sec. 4.4.3)
dlsReductionFactor_II	0,5	0,5	0,5	0,5	0,5	0,5	0,5	0,5	0,5	0,5	0,5	0,5	DLS reduction factor EN 1998-1, Sec. 4.4.3)
dlsReductionFactor_III	0,4	0,4	0,4	0,4	0,4	0,4	0,4	0,4	0,4	0,4	0,4	0,4	DLS reduction factor EN 1998-1, Sec. 4.4.3)
dlsReductionFactor_IV	0,4	0,4	0,4	0,4	0,4	0,4	0,4	0,4	0,4	0,4	0,4	0,4	DLS reduction factor EN 1998-1, Sec.

													4.4.3)
bendDriftOM_NCR_EN8_1_Sec_C4_2_1	0,008	0,008	0,008	0,008	0,008	0,008	0,008	0,008	0,008	0,008	0,008	0,008	EN 1998-3, Bending capacity for ordinary masonry at NCR according to EN 1998-3, C.4.2.1
shearDriftOM_NCR_EN8_1_Sec_C4_3_1	0,004	0,004	0,004	0,004	0,004	0,004	0,004	0,004	0,004	0,004	0,004	0,004	EN 1998-3, Shear capacity for ordinary masonry at NCR according to EN 1998-3, C.4.2.1
damageDriftOM_DLR_EN8_1_Sec	0,005	0,005	0,005	0,005	0,005	0,005	0,005	0,005	0,005	0,005	0,005	0,005	Drif limit for DLR in Ordinary Masonry EN 1998-1 4.4.3.2
bendDriftRM_NCR_EN8_1_Sec_C4_2_1	0,008	0,008	0,008	0,008	0,008	0,008	0,008	0,008	0,008	0,008	0,008	0,008	EN 1998-3, Bending capacity for reinforced masonry at NCR according to EN 1998-3, C.4.2.1
shearDriftRM_NCR_EN8_1_Sec_C4_3_1	0,004	0,004	0,004	0,004	0,004	0,004	0,004	0,004	0,004	0,004	0,004	0,004	EN 1998-3, Shear capacity for reinforced masonry at NCR according to EN 1998-3, C.4.2.1
damageDriftRM_DLR_EN8_1_Sec	0,005	0,005	0,005	0,005	0,005	0,005	0,005	0,005	0,005	0,005	0,005	0,005	Drif limit for DLR in Reinforced Masonry EN 1998-1 4.4.3.2
bendDriftCM_NCR_EN8_1_Sec_C4_2_1	0,008	0,008	0,008	0,008	0,008	0,008	0,008	0,008	0,008	0,008	0,008	0,008	EN 1998-3, Bending capacity for confined masonry at NCR according to EN 1998-3, C.4.2.1
shearDriftCM_NCR_EN8_1_Sec_C4_3_1	0,004	0,004	0,004	0,004	0,004	0,004	0,004	0,004	0,004	0,004	0,004	0,004	EN 1998-3, Shear capacity for confined masonry at NCR according to EN 1998-3, C.4.2.1
damageDriftCM_DLR_EN8_1_Sec	0,005	0,005	0,005	0,005	0,005	0,005	0,005	0,005	0,005	0,005	0,005	0,005	Drif limit for DLR in Confined Masonry EN 1998-1 4.4.3.2
bendDriftRC_NCR_EN8_1_Sec_C4_2_1	0,011	0,011	0,011	0,011	0,011	0,011	0,011	0,011	0,011	0,011	0,011	0,011	EN 1998-3, Bending capacity for confined masonry at NCR according to EN 1998-3, C.4.2.1
shearDriftRC_NCR_EN8_1_Sec_C4_3_1	0,011	0,011	0,011	0,011	0,011	0,011	0,011	0,011	0,011	0,011	0,011	0,011	EN 1998-3, Shear capacity for confined masonry at NCR according to EN 1998-3, C.4.2.1
damageDriftRC_DLR_EN8_1_Sec	0,005	0,005	0,005	0,005	0,005	0,005	0,005	0,005	0,005	0,004	0,005	0,005	Drif limit for DLR in Reinforced Concrete EN 1998-1 4.4.3.2
ceilingLoadDeadLoad	0	0	0	0	0	0	0	0	0	0	0	0	Default ceiling dead load

ceilingLoadPermanentLoad	2,5	1,2	1,2	1,2	1,5	1,2	1,2	1,2	1,2	1,2	1,2	1,2	Default ceiling permanent load
ceilingLoadPermanentGammaG	1	1	1	1	1	1	1	1	1	1	1	1	Load coeff. for permanent load in static analysis, gamma_G
ceilingLoadPermanentCoefStatic	1,35	1,35	1,35	1,35	1,35	1,35	1,35	1,35	1,35	1,35	1,35	1,35	
ceilingLoadLifeLoad	2	2,5	2,5	2,5	2	2,5	2,5	2,5	2,5	2,5	2,5	2,5	Default ceiling life load
ceilingLoadLifeLoadPOPhi	1	0,8	0,8	0,8	0,8	0,8	0,8	0,8	0,8	0,8	0,8	0,8	Load coeff. for life load for pushover, Phi (see EN 1998-1, Table 4.2)
ceilingLoadLifeLoadPOKsi2	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	Load coeff. for life load for pushover, Ksi_2i (see EN 1990, Table A1.1)
ceilingLoadLifeLoadGammaQ	1,5	1,5	1,5	1,5	1,5	1,5	1,5	1,5	1,5	1,5	1,5	1,5	Load coeff. for life load, gamma_Q
ceilingLoadSnowLoad	0	0	0	0	0	0	0	0	0	0	0	0	Default roof snow load
ceilingLoadSnowLoadPOPhi	1	1	1	1	1	1	1	1	1	1	1	1	Load coeff. for snow load for pushover, Phi (see EN 1998-1, Table 4.2)
ceilingLoadSnowLoadPOKsi2	0,2	0,2	0,2	0	0,2	0,2	0,2	0,2	0,2	0,2	0,2	0	Load coeff. for snow load for pushover, Ksi_2i (see EN 1990, Table A1.1), for H>1000 m
ceilingLoadSnowLoadGammaQ	1,5	1,5	1,5	1,5	1,5	1,5	1,5	1,5	1,5	1,5	1,5	1,5	Load coeff. for snow load, gamma_Q, used for static analysis
ceilingLoadSnowLoadKsi0	0,7	0,7	0,7	0,5	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,5	Combination factor for snow load for H>1000 m for static analysis
reinforcementDefaultFyk	550	500	500	500	500	500	500	500	355	355	500	500	Default reinforcement yield strength, characteristic
reinforcementDefaultDiameter	10	10	10	10	10	10	10	10	10	10	10	10	Default reinforcement diameter
reinforcementDefaultModulE	210000	210000	210000	210000	200000	210000	210000	210000	210000	210000	210000	210000	Default reinforcement modulus, mean
reinforcementDefaultWeight	7800	7800	7800	7800	7800	7800	7800	7800	7800	7800	7800	7800	Default reinforcement weight, kg/m3
reinforcementDefaultFykFym	0,91	0,91	0,91	0,98	0,91	0,91	0,91	0,91	0,91	0,91	0,91	0,98	Default reinforcement ratio of char/mean strength

rho_n	0,75	0,75	0,75	0,75	5	0,75	0,75	0,75	0,75	0,75	0,75	0,75	Rho_n according 5.5.1.2
rho_t	1	1	1	1	10	1	1	1	1	1	1	1	Rho_t according 5.5.1.3
def_fb_horizontal_Group2	2	2	2	2	2	2	2	2	2	2	2	2	default brick strength in horizontal direction for Group 2
def_fb_horizontal_Group3	1,4	1,4	1,4	1,4	1,4	1,4	1,4	1,4	1,4	1,4	1,4	1,4	default brick strength in horizontal direction for Group 3
k_Mult_Hor_Strength	0,5	0,5	0,5	0,5	0,5	0,5	0,5	0,5	0,5	0,5	0,5	0,5	multiplier for coeff. K for the calculation of horizontal wall strength f _k
alfa_Table_3_3_EN6_1_1_e1_1	0,65	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	power for fb in equation 3.2 - 3.4 in EN 1996-1-1
alfa_Table_3_3_EN6_1_1_e2_1	0,65	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	power for fb in equation 3.2 - 3.4 in EN 1996-1-1
alfa_Table_3_3_EN6_1_1_e3_1	0,65	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	power for fb in equation 3.2 - 3.4 in EN 1996-1-1
alfa_Table_3_3_EN6_1_1_e4_1	0,65	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	power for fb in equation 3.2 - 3.4 in EN 1996-1-1
alfa_Table_3_3_EN6_1_1_e1_2	0,7	0,85	0,85	0,85	0,85	0,85	0,85	0,85	0,85	0,7	0,85	0,85	power for fb in equation 3.2 - 3.4 in EN 1996-1-1
alfa_Table_3_3_EN6_1_1_e2_2	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	power for fb in equation 3.2 - 3.4 in EN 1996-1-1
alfa_Table_3_3_EN6_1_1_e3_2	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	power for fb in equation 3.2 - 3.4 in EN 1996-1-1
alfa_Table_3_3_EN6_1_1_e4_2	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	power for fb in equation 3.2 - 3.4 in EN 1996-1-1
alfa_Table_3_3_EN6_1_1_e1_3	0,65	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	power for fb in equation 3.2 - 3.4 in EN 1996-1-1
alfa_Table_3_3_EN6_1_1_e2_3	0,65	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	power for fb in equation 3.2 - 3.4 in EN 1996-1-1
alfa_Table_3_3_EN6_1_1_e3_3	0,65	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	power for fb in equation 3.2 - 3.4 in EN 1996-1-1
alfa_Table_3_3_EN6_1_1_e4_3	0,65	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	0,7	power for fb in equation 3.2 - 3.4 in EN 1996-1-1

beta_Table_3_3_EN6_1_1_e1_1	0,25	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	power for fm in equation 3.2-3.4 in EN 1996-1-1
beta_Table_3_3_EN6_1_1_e2_1	0,25	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	power for fm in equation 3.2-3.4 in EN 1996-1-1
beta_Table_3_3_EN6_1_1_e3_1	0,25	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	power for fm in equation 3.2-3.4 in EN 1996-1-1
beta_Table_3_3_EN6_1_1_e4_1	0,25	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	power for fm in equation 3.2-3.4 in EN 1996-1-1
beta_Table_3_3_EN6_1_1_e1_2	0	0	0	0	0	0	0	0	0	0	0	0	power for fm in equation 3.2-3.4 in EN 1996-1-1
beta_Table_3_3_EN6_1_1_e2_2	0	0	0	0	0	0	0	0	0	0	0	0	power for fm in equation 3.2-3.4 in EN 1996-1-1
beta_Table_3_3_EN6_1_1_e3_2	0	0	0	0	0	0	0	0	0	0	0	0	power for fm in equation 3.2-3.4 in EN 1996-1-1
beta_Table_3_3_EN6_1_1_e4_2	0	0	0	0	0	0	0	0	0	0	0	0	power for fm in equation 3.2-3.4 in EN 1996-1-1
beta_Table_3_3_EN6_1_1_e1_3	0,25	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	power for fm in equation 3.2-3.4 in EN 1996-1-1
beta_Table_3_3_EN6_1_1_e2_3	0,25	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	power for fm in equation 3.2-3.4 in EN 1996-1-1
beta_Table_3_3_EN6_1_1_e3_3	0,25	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	power for fm in equation 3.2-3.4 in EN 1996-1-1
beta_Table_3_3_EN6_1_1_e4_3	0,25	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	0,3	power for fm in equation 3.2-3.4 in EN 1996-1-1