

**BUILDING INFORMATION MODELLING IN STRUCTURAL
ANALYSIS**

Finite element method



Bachelor's thesis

Construction Engineering

Visamäki, spring 2020

Ilja Loginov

Degree Programme in Construction Engineering
Hämeenlinna University Centre

Author	Ilja Loginov	Year 2020
Title	Building information modelling in structural analysis	
Supervisor(s)	Cristina Tirteu	

ABSTRACT

The purpose of this Bachelor's thesis was to examine the applicability of Dlubal RFEM, a structural analysis software based on Finite Element method on a renovation project made in a BIM software Autodesk Revit. The aim was to make observations on the performance and collaboration of the software and produce useful information about learning to use it.

The thesis consisted of making a virtual structural model of a residential multi-storey building's frame and using it to generate a modern alternative to the 100-year-old structure with minimum changes made to the historical and architectural appearance of the building. The model was then subjected to a structural analysis where suggestions were made regarding its structural design. Throughout the project, observations were made on both Autodesk Revit and Dlubal RFEM features and its user assistance materials.

As a result of the thesis it was concluded that Dlubal RFEM can be practical in combination with Autodesk Revit for such renovation projects, even though greatly affecting performance of computer the generated materials satisfy initial requirements and provide new opportunities for the further steps of the project.

Keywords BIM, CAD, FEM, FEA, renovation, Autodesk Revit, Dlubal RFEM.

Pages 57 pages including appendices 28 pages

CONTENTS

1	INTRODUCTION	1
2	BUILDING INFORMATION MODELLING	1
3	AUTODESK REVIT	2
4	DLUBAL RFEM	3
5	AS OY FRANZENINAUKIO PROJECT	4
5.1	Building information	4
5.1.1	Demolition of old structures	6
5.1.2	Intermediate floor repairs	6
6	BUILDING MODEL	8
6.1	Model preparation	8
6.2	Structural modelling	10
6.3	Exporting the model	11
7	STRUCTURAL ANALYSIS	12
7.1	Conflicts after transfer	12
7.2	International technical standards	14
7.3	Loads and Load combinations	15
7.4	Finite element analysis	18
8	STRUCTURAL ANALYSIS RESULTS	19
8.1	Result summary	19
8.2	Deformations	20
8.3	Support reactions	21
8.4	Internal forces	22
8.5	Structural components design	22
8.5.1	RF-CONCRETE	22
8.6	Updating existing BIM	24
9	CONCLUSION	24
	REFERENCES	26

- Appendix 1 Old demolished, damaged and repaired floor slabs
- Appendix 2 Autodesk Revit generated model and drawings
- Appendix 3 Dlubal RFEM generated analytical model
- Appendix 4 Structural analysis results
- Appendix 5 RF-Concrete structural designs
- Appendix 6 Updated BIM Model

1 INTRODUCTION

With the rising number of buildings approaching the end of their serviceability period, renovation plays an ever-larger role in residential and commercial expansions. The goal of this Bachelor's thesis is to apply structural renovation on a multi-storey building to increase its sustainability and serviceability period, while preserving its cultural and architectural appearance. This is a stage where Building Information Modelling (BIM) plays a significant role in the construction process.

There are many BIM platforms in use today, each coming with its pros and cons. This thesis specifically explores Autodesk Revit, a powerful BIM software, and Dlubal RFEM, an easy yet powerful structural analysis program based on Finite Element method (FEM).

The thesis shows the process of structural modelling applied on an existing aged building in order to assist its potential future renovation. Additionally, it explores renovation possibilities, performance, application and collaboration of software providing insights into learning to use Autodesk Revit and Dlubal RFEM that can be useful to other people working or studying in construction industry.

This is done through observation and application of tools provided by the software. The programs are used to generate data on buildings and make suggestions on its structural design and potential renovation.

The building is of cultural and historical value located in Kallio, Helsinki. It is a five-storey residential building with an additionally constructed sixth floor in the old attic area. The goal of this project is to perform a structural renovation by replacing the old deteriorating intermediate concrete floor with a new reinforced concrete structure to prolong serviceability of the building.

2 BUILDING INFORMATION MODELLING

Building Information Modelling (BIM) can be depicted as a process of developing, managing and bringing into existence a digital project, which contains functional characteristics and physical properties of purposefully used facilities. Based on the results of extensive research the outcome of this process is represented in a model which nowadays is generated through a specially developed software. Apart from graphical visualisation a model includes a decisive information concerning a project cycle and every aspect of it (Cherkaoui, 2017).

Rapidly growing due to its applicability, BIM technology has become a widely discussed topic and an essential part of construction industry. Possibility to visualize and alter the existent and non-existent facilities has allowed project managers to see a construction prior its physical execution, reduce conflicting areas of the project, improve safety issues, and analyse and apply simulations of potential impacts. Additionally, it allows controlling estimation, costs, material supply and facility maintenance throughout the lifespan of a project, affecting its efficiency (Cherkaoui, 2016).

Playing a major role in BIM, Computer-Aided Design (CAD) has granted people the possibility to represent and record the shape information, which can be altered at any time, given the possession of the BIM model. That made an impact on visual communication and allowed people to learn about the progression of the erection, functionality of building components and required quantity of materials regarding an upcoming project. Instantaneous modifications to the model have preserved large amount of time spent by drafters to apply the necessary changes. It significantly raised accuracy of the drawings and production of the drafting copies (Cherkaoui, 2017).

With a wide variety of BIM platforms present today, each coming with its assets and liabilities, the choice falls on specific tools, provided by the software. Primarily used in architecture, engineering and construction (AEC), BIM software unites all three disciplines in to one industry. Integration of the separate parts of one project and working together to bring it into existence has made work and collaboration more efficient.

3 AUTODESK REVIT

Autodesk, Inc. is a multinational software corporation founded in 1982 in the United States by John Walker. It specializes in several business fields: product design, manufacturing, architecture, engineering, construction, media, and entertainment (Funding Universe, n.d.). With the company's main office located in San Rafael, California, Autodesk is a leader in 3D design and engineering, having customers worldwide and offices established in more than thirty countries (Autodesk, 2019).

The company is best known for their CAD platforms, AutoCAD – a drafting application, and Revit – a powerful BIM software. Both are widely used by architects, engineers and structural designers to design, draft and model structures and other different objects.

Autodesk Revit was originally developed by Charles River Software, founded in 1997 by Leonid Raiz and Irwin Jungreis. It was later renamed in Revit Technology Corporation in 2000 and acquired by Autodesk in 2002.

Intending to create an architectural version of software that could handle more complex projects than ArchiCAD, Revit came out as a very powerful BIM collaboration tool, which brings the unique perspectives of different disciplines into one three-dimensional model, including geometry, design and construction information (Bergin, 2012). The interface of the program can be seen in Figure 1.

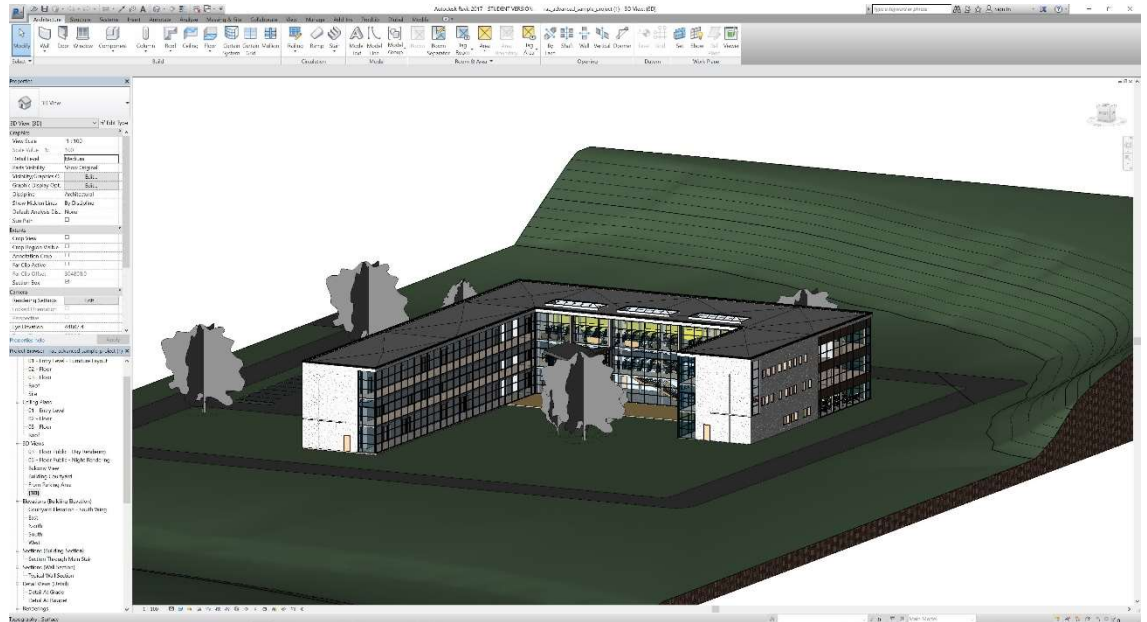


Figure 1. Interface screenshot of Autodesk Revit

4 DLUBAL RFEM

DLUBAL software was founded as a one-man company in 1987 in Harswinkel, Germany by George Dlubal. The company is specialized in structural analysis solutions which are based on FEM. The method implies approximation of the unknown result by subdividing a large system into smaller simpler parts, running them through algebraic equations and assembling into a larger system, modelling the entire problem. Some of the interest areas of the problems are structural analysis, heat transfer, fluid flow and electromagnetic potential (Finite element method (FEM), 2019).

Ever since its establishment the company has continuously developed and optimized user-friendly and powerful programs for structural design and analysis. The most notable ones are RSTAB – an analysis and design software for beam, frame and truss structures, and RFEM – a 3D Finite Element Analysis (FEA) software for design and analysis of structures made of various materials.

Having more than 200 employees the company have established offices in 6 countries at 9 locations and is being used by more than 7800 companies

with more than 45,000 satisfied customers in 95 countries (Dlubal software, 2019).

Dlubal RFEM is a 3D FEA software operating under Microsoft Windows. The program is used to define structures, various materials such as concrete, steel, timber, glass, and various loads. RFEM consists of different corresponding add-on modules providing deformations, internal forces, stresses, support forces and soil contact stresses. Such concept allows to manually tailor a program package that suits a user's individual needs and allows to perform further analysis and design according to various standards (RFEM, 2019). The interface of RFEM is shown below in Figure 2.

The program offers numerous interfaces and a wide range of useful features for data exchange within the BIM process, maintaining all data in a digital 3D model, which is used throughout all planning stages. As a result, RFEM is a good alternative to other structural analysis software such as SAP 2000, RISA and Robot Structural analysis, which has an integration in the standard BIM software such as Revit or Tekla. Various CAD and structural analysis programs can use the same model, which can be transferred between programs.

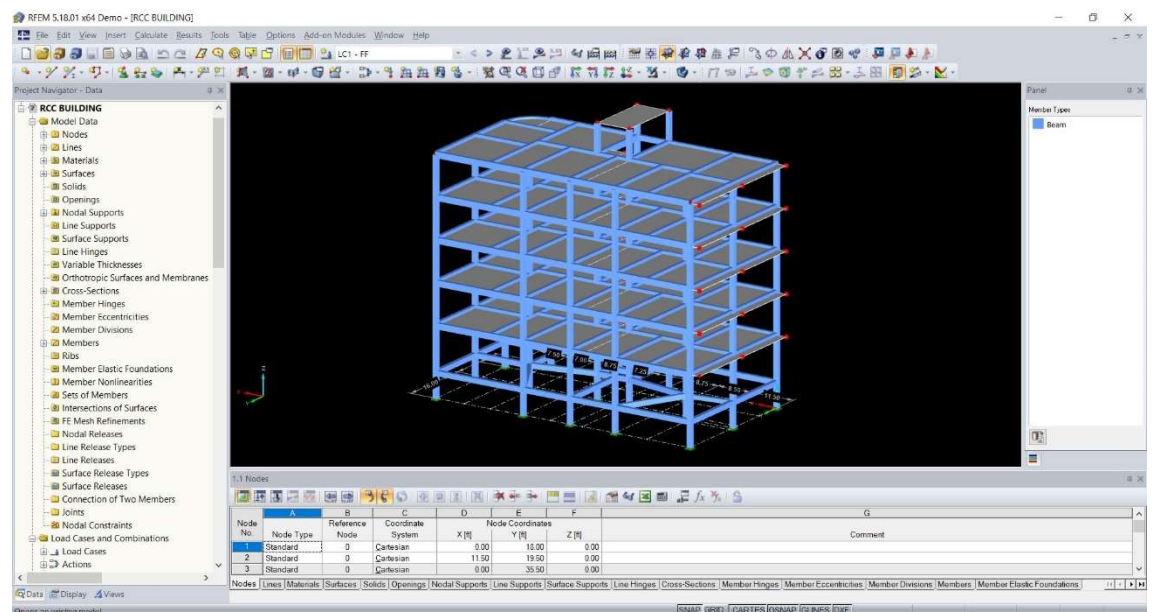


Figure 2. Interface screenshot of DLUBAL RFEM

5 AS OY FRANZENINAKIO PROJECT

5.1 Building information

As Oy Helsingin Franzeninakuo is a multi-storey building located on Fleminginkatu 10, Kallio, Helsinki. The original designs of the project are

dated in 1910 and the building itself was erected in 1912 by the Finnish designer and builder Heikki Kaartinen, (Figure 3). The building is of great architectural, cultural and historical value and falls under a protection class Sr-2 (suojeltava rakennus), which restricts demolition, extension and reconstruction works of any parts of the building that affect historical value or architectural features without permission from the building committee. The building has four residential floors and an attic floor. Two spacious staircases going through the building.



Figure 3. Fleminginkatu 10. Franzeninkatu 23 (Helsinki kuvia, n.d.)

During a major renovation taking place in 1980-1982, two elevator shafts were constructed on both sides of staircases, going from the ground floor to the last floor of the residential area. The elevator and ventilation engine rooms were placed on the attic. The basement is redesigned for office and storage area and cold offices in the attic were replaced with additional storage area. Improvements were made to electricity, plumbing and ventilation allowing extension of the bathrooms. During the renovation, the style of 1910s faced, roof structure, spacious stairwells and the heights of the apartments were preserved (Rakennuslupa, 2016).

A major renovation took place in 2016-2018 with major changes done to the deteriorating wooden roof structure and apartment arrangement. Previously installed electrical and HVAC systems were completely removed and replaced with a modern one. Old and deteriorating exterior plaster and ornament was removed and reapplied anew with the same architectural view as the original (Rakennuslupa, 2016).

5.1.1 Demolition of old structures

Renovation included construction of eight new residential apartments on the attic floor, for which to be feasible the old roof structure had to be demolished and rebuilt anew, differing as little as possible from the original architectural view. Though majority of old chimneys were preserved, slight modifications were made to them to fit the ventilation pipes. The demolition drawing of the attic can be found in Appendix 2.

The old wooden attic and residential intermediate floors were removed and emptied of the old insulation, revealing the original structure (see Figure 4). To bear the new roof structure, additional beams were placed between the structure's external and internal bearing walls and overlapping concrete slabs. The internal part of the eaves is reinforced to support the bottom of the new roof. The old staircase floor of the attic area is demolished and cast anew to match the level of the attic's new intermediate floor.



Figure 4. Demolished roof structure

5.1.2 Intermediate floor repairs

As the old wooden floor structure was removed, it was revealed that the old intermediate concrete slabs were covered with cracks, going from top to bottom of the slab at some places. Holes in some slabs also showed the

lack of reinforcement in them (see Figure 5). The slabs themselves rested on the bottom eaves of the old INP 200 beams.



Figure 5. Cracked floor slab

Since renovation required removal of the old wool, filled in the area between the slab and wooden floor, and refilling the empty space with a heavier small grain gravel, it was necessary to perform an inspection for the presence of cracks on all the concrete slabs and make necessary repairs. Old demolished structures, damaged and repaired slabs can be seen on the photos given in Appendix 1.

As the minor reparations were made to the concrete slabs, leaving uncracked ones under heavier load, it is certain that more cracking will occur, and the building will be a subject to a major structural renovation after next serviceability check (see Figure 6).



Figure 6. Repaired intermediate floor slab

6 BUILDING MODEL

6.1 Model preparation

The generation of the models in Autodesk Revit is done using the parametric modelling method, which means that the model is created based on a series of pre-programmed algorithms. Using its internal logic, the program is automatically generating elements rather than manually manipulating them (BIM Wiki, 2019). Autodesk offers numerous useful and easy to follow video lectures and tutorials to assist users in operating their programs.

The file containing original 3D BIM model is generated using architectural template in Autodesk Revit (Figure 7). It includes both architectural and structural drawings, types of structures, properties of objects and

installation guidance. The list of attached drawings can be seen in Table 1, parts of which can be found in Appendix 2.

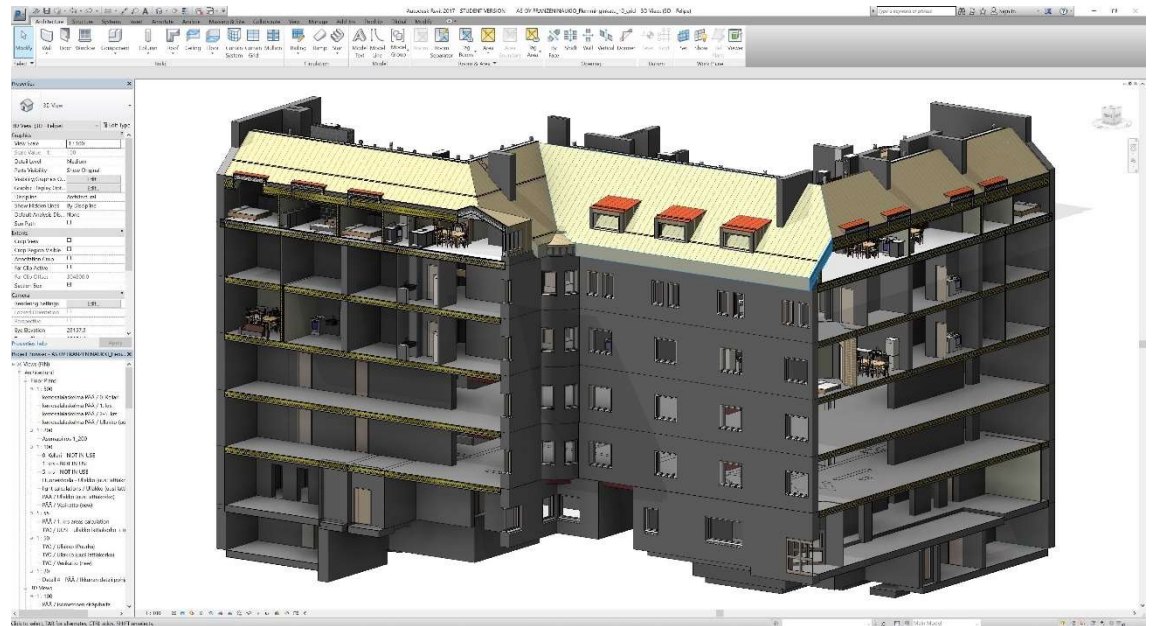


Figure 7. As Oy Franzeninaukio 3D architectural model

Table 1. List of attached architectural and structural drawings

A	B	C	D
Sheet Category	Sheet Sub-1 category	Sheet Number	Sheet Name
01 Architectural			
02 Asiakirjaluettelo			
01 Architectural	02 Asiakirjaluettelo	100 01	AR piirustusluettelo
03 Pääpiirustus			
01 Architectural	03 Pääpiirustus	002 01	ASEMAPIIRROS
01 Architectural	03 Pääpiirustus	004 01	LEIKKAUS A-A JA PIHJULKISIVU
01 Architectural	03 Pääpiirustus	005 01	JULKISIVU FRANZENINKATU
01 Architectural	03 Pääpiirustus	003 01	POHJAPIRUSTUS ULLAKKO
01 Architectural	03 Pääpiirustus	004 03	LEIKKAUS C-C JA PIHJULKISIVU
01 Architectural	03 Pääpiirustus	004 02	LEIKKAUS B-B
01 Architectural	03 Pääpiirustus	005 02	JULKISIVU FLEMIGINKATU
01 Architectural	03 Pääpiirustus	009 01	RAKENNETYYPIT
01 Architectural	03 Pääpiirustus	003 02	POHJAPIRUSTUS VESIKATTO
01 Architectural	03 Pääpiirustus	004 04	LEIKKAUKSET DD, EE JA FF
01 Architectural	03 Pääpiirustus	009 02	KATTOIKKUNAT
01 Architectural	03 Pääpiirustus	LIITE 1	VESIKATON VISUALISOINTI
01 Architectural	03 Pääpiirustus	LIITE 2	PINTA-ALA KAAVIOT
01 Architectural	03 Pääpiirustus	002 02	PIHAPIRUSTUS
01 Architectural	03 Pääpiirustus	LIITE 4	ASEMAPIIRROS
04 Urakkalaskenta			
01 Architectural	04 Urakkalaskenta	103 01	Pohjapiirustus ullakko
01 Architectural	04 Urakkalaskenta	103 02	Pohjapiirustus vesikatto
01 Architectural	04 Urakkalaskenta	103 01P	Purkupiirustus ullakko
01 Architectural	04 Urakkalaskenta	109 01	Kattoikkunat
01 Architectural	04 Urakkalaskenta	109 02	Kattolohvit
01 Architectural	04 Urakkalaskenta	109 04	Rakennetwoit
01 Architectural	04 Urakkalaskenta	109 03	Ikkuna kaavio
01 Architectural	04 Urakkalaskenta	109 05	Ovuluettelo - Ullakko
02 Structural			
02 Structural	BIM	400	PIRUSTUSLUETTELO
Tvopiirustus			
02 Structural	Tvopiirustus	310	Rakennetwoit
02 Structural	Tvopiirustus	304	ULLAKON VALIPOHJA
02 Structural	Tvopiirustus	305	YLÄPOHJA YLÄPOHJA - KANTAVAT
02 Structural	Tvopiirustus	311	LEIKKAUS 1 - 1 AUKKOPALKKI
02 Structural	Tvopiirustus	312	LEIKKAUS 2 - 2 YLEISLEIKKAUS
02 Structural	Tvopiirustus	315	LEIKKAUS 5 - 5 TORNI
02 Structural	Tvopiirustus	317	LEIKKAUS 7 - 7 HORMILÄPIENNIT
02 Structural	Tvopiirustus	314	LEIKKAUS 4 - 4 LAPEIKKUNA HVS
02 Structural	Tvopiirustus	313	LEIKKAUS 3 - 3 LAPEIKKUNA
02 Structural	Tvopiirustus	321	DET 1-1
02 Structural	Tvopiirustus	319	LEIKKAUS 9 - 9 MÄRKÄTILA
02 Structural	Tvopiirustus	320	LEIKKAUS 10 - 10 JIIRI
02 Structural	Tvopiirustus	316	LEIKKAUS 6 - 6 KATTOIKKUNA PERIA
02 Structural	Tvopiirustus	322	DET 2-2
02 Structural	Tvopiirustus	323	DET 3-3
02 Structural	Tvopiirustus	324	DET 4-4
02 Structural	Tvopiirustus	325	DET 5-5
02 Structural	Tvopiirustus	326	DET 6-6
02 Structural	Tvopiirustus	327	DET 7-7
02 Structural	Tvopiirustus	328	DET 8-8
02 Structural	Tvopiirustus	329	DET 9-9
02 Structural	Tvopiirustus	331	DET 11-11
02 Structural	Tvopiirustus	332	DET 12-12
02 Structural	Tvopiirustus	333	ULLAKON PURKUPIRUSTUS
02 Structural	Tvopiirustus	318	LEIKKAUS 8 - 8 PORRASKAYTÄVAN
02 Structural	Tvopiirustus	309	ULLAKKO, PALOKATKOT
02 Structural	Tvopiirustus	330	DET 10-10
02 Structural	Tvopiirustus	335	PALOKATKODETALJIT

Following the given structural drawings, the building is retraced on the structural template, as seen in Figure 8. All objects related to architectural representation are removed, changing architecturally generated structural objects to analytical structural objects. This is done to obtain a simplified analytical model of the building which is used for structural analysis. Revit provides various properties applicable to any generated objects as well as a great choice of structural elements, which can be withdrawn from the library or generated manually.

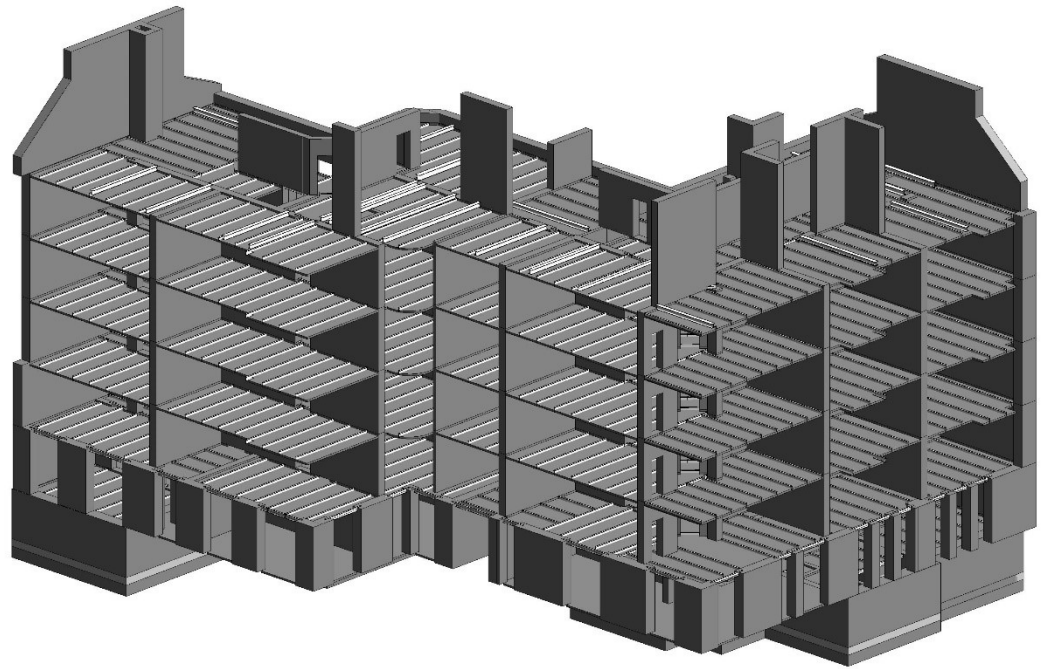


Figure 8. 3D structure of the old building

6.2 Structural modelling

Due to the failure of the old concrete floors and stress on the bricks from the edges of the INP200 beams, the old intermediate floor structure is removed in this project and a more durable supporting frame added to the building, to increase its sustainability (see Figure 9). The frame consists of 380x380mm reinforced concrete columns, 300x300mm reinforced concrete beams and 150mm reinforced concrete floor.

To aid reinforcing process Revit provides a variety of stiffening shapes for presentation. Stiffening is applied approximately during the design stage of the project; it is afterwards analysed with the whole building and corrections are made in case of failure of the element.

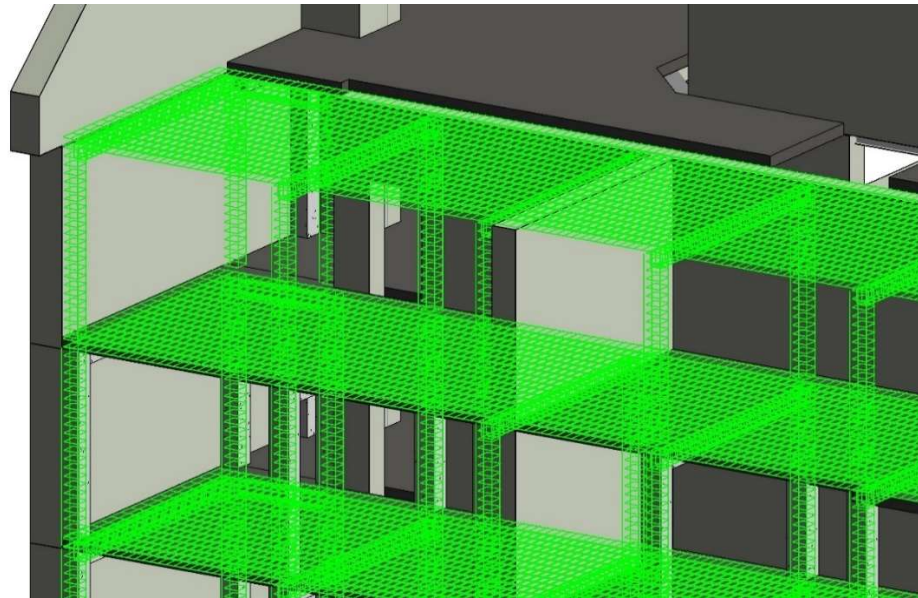


Figure 9. Generated supporting frame

6.3 Exporting the model

The prepared analytical model is displayed in an easy and comprehensive building structure which includes physical and analytical properties of elements in detail, as seen in Figure 10 on the next page. Prior structural analysis, the model is exported to RFEM. If both programs are installed, Revit makes the access of RFEM's interface directly available through its interface.

The elements and their properties can be manually selected or limited to the model and/or load data in exporting properties (see Figure 11). This is useful when defined elements are set in an external program, but only specific properties are needed be imported into the original program.

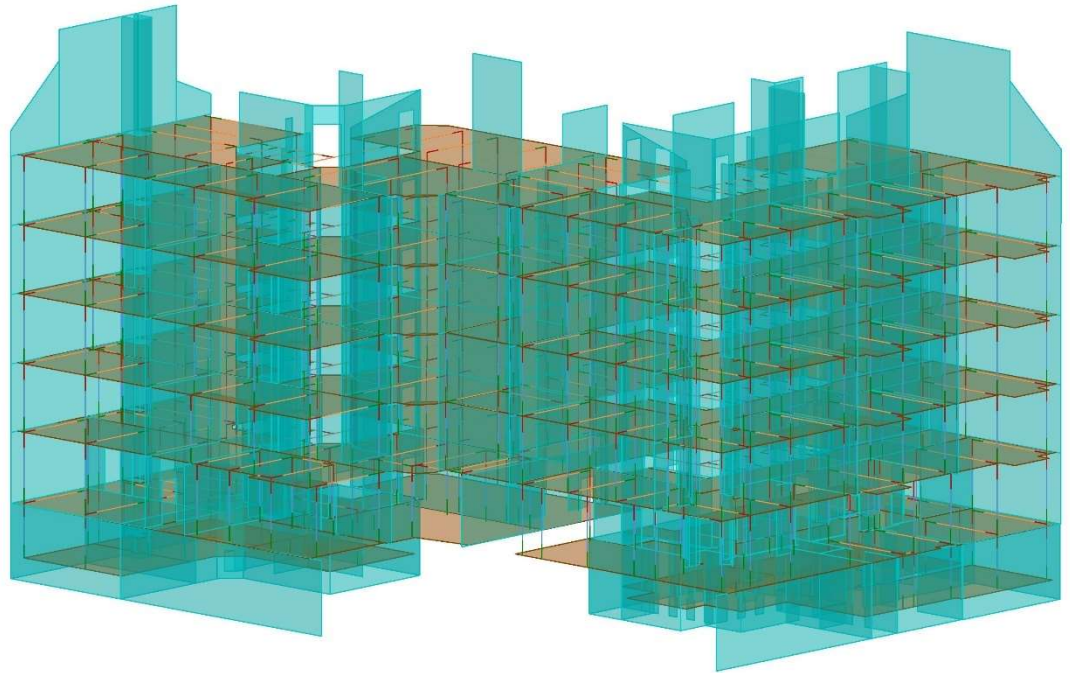


Figure 10. Analytical model of the building

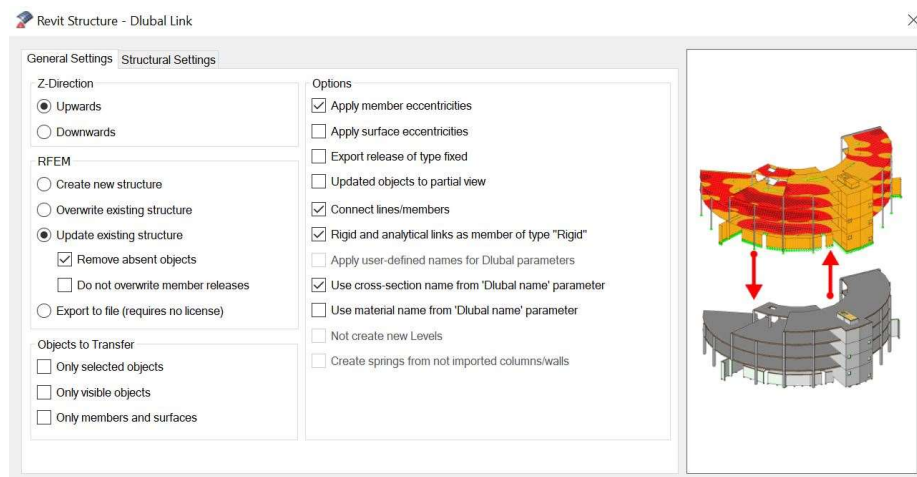


Figure 11. Revit Structure – Dlubal Link

7 STRUCTURAL ANALYSIS

7.1 Conflicts after transfer

Unlike Revit, RFEM uses Object-oriented programming (OOP) a concept of objects, which contain data and code in the form of procedures (Technopedia, 2012). During the export the existing RFEM data is checked and information is added. An equivalent object is added for beam, column, floor or wall, not just collection of lines and surfaces. Due to adjustable reference lines, the load bearing elements are merged in an optimal calculation model, connecting centre points of intersecting beams and columns. The exported model is presented in Figure 12.

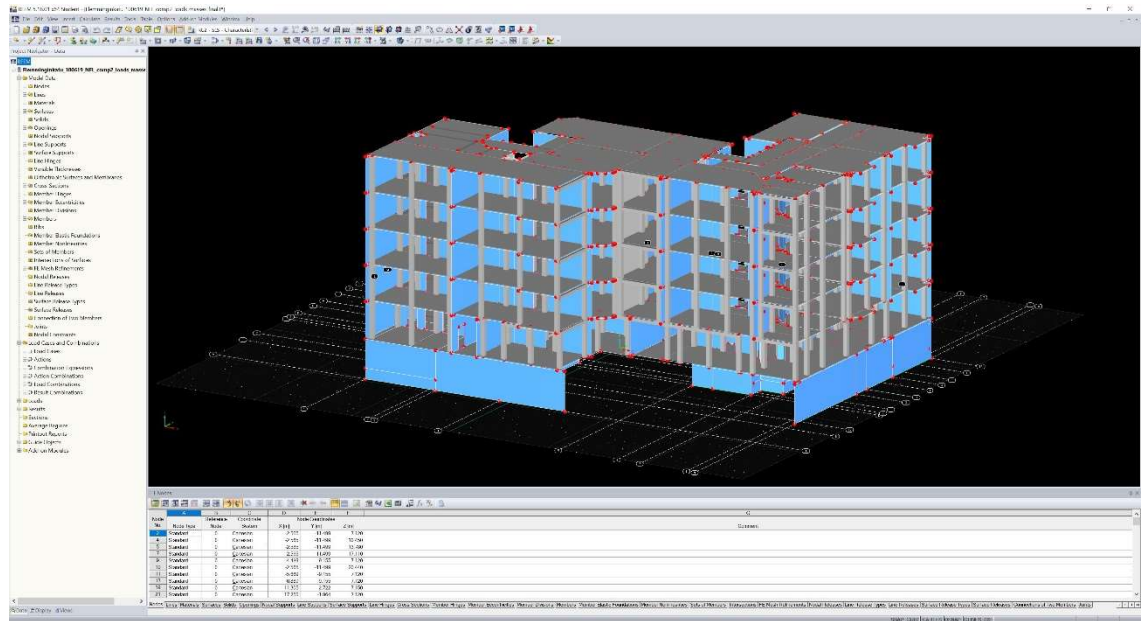


Figure 12. RFEM analytical model

Prior to running the structural analysis on the building, the model is subjected to issue checks. The model check allows to search and reveal any occurring problems or conflicts, be that identical nodes, overlapping lines, members or surfaces, or crossing unconnected lines and members. The problems are shown on the model, as seen in Figure 13 and then are corrected manually. The plausibility check of entries is done to see if calculation would be feasible.

Structures consisting of steel, reinforced concrete and timber can be designed using tools provided by RFEM. The list of materials is found in Materials table and is marked with a comment if the material was imported or used from a programs database. Material table can be seen in Figure 14.

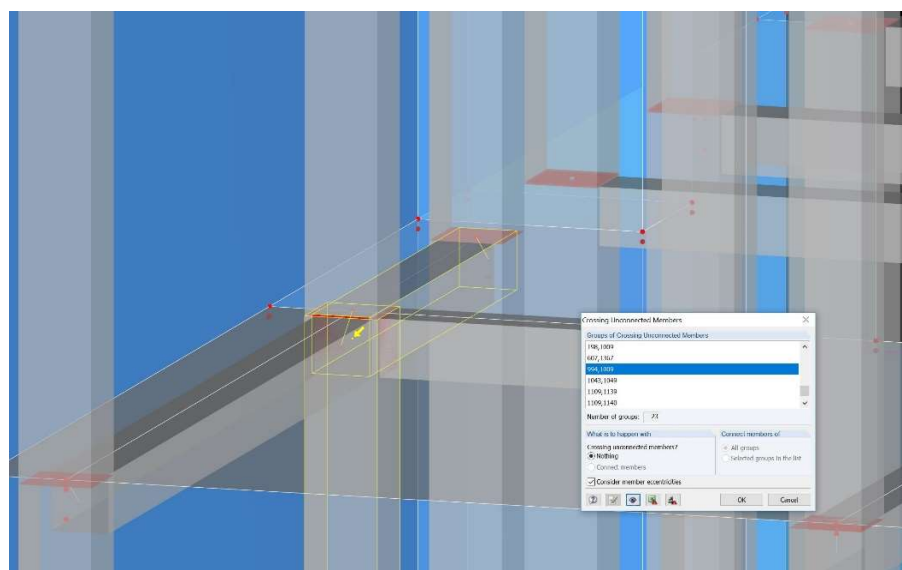


Figure 13. Crossing unconnected members

1.3 Materials

Material No.	A Material Description	B Modulus of Elasticity E [kN/cm ²]	C Shear Modulus G [kN/cm ²]	D Poisson's Ratio ν [-]	E Specific Weight γ [kN/m ³]	F Coeff. of Th. Exp. α [1/°C]	G Partial Factor γ _M [-]	H Material Model	I Comment
2	Concrete C25/30 EN 1992-1-1:2004/A	3100.00	1291.67	0.200	25.00	1.00E-05	1.00	Isotropic Linear Elastic	Revit material constants
3	Concrete C25/30 EN 1992-1-1:2004/A	3100.00	1291.67	0.200	25.00	1.00E-05	1.00	Isotropic Linear Elastic	Revit material constants
4	Concrete C30/37 EN 1992-1-1:2004/A	3300.00	1375.00	0.200	25.00	1.00E-05	1.00	Isotropic Linear Elastic	Revit material constants
5	Masonry	2325.00	996.14	0.167	24.07	1.00E-05	1.00	Isotropic Linear Elastic	Revit material constants
6									
7									
8									
9									
10									
11									
12									
13									
14									

Nodes | Lines | Materials | Surfaces | Solids | Openings | Nodal Supports | Line Supports | Surface Supports | Line Hinges | Cross-Sections | Member Hinges | Member Eccentricities | Member Divisions | Members

Figure 14. Material table of exported building

7.2 International technical standards

Dlubal RFEM implements numerous international technical standards describing different design situations. These situations determine the expected conditions of the building during its construction and usage. The following standards amongst others include Eurocodes, American Standards, Russian Standards, Chinese Standards, Indian Standards and Brazilian Standards. Load and result combinations generated on the analytical model during this project are done according to SFS-EN 1990 Eurocode – Basis of structural design. Information about design situation is shown in Figure 15 and Figure 16.

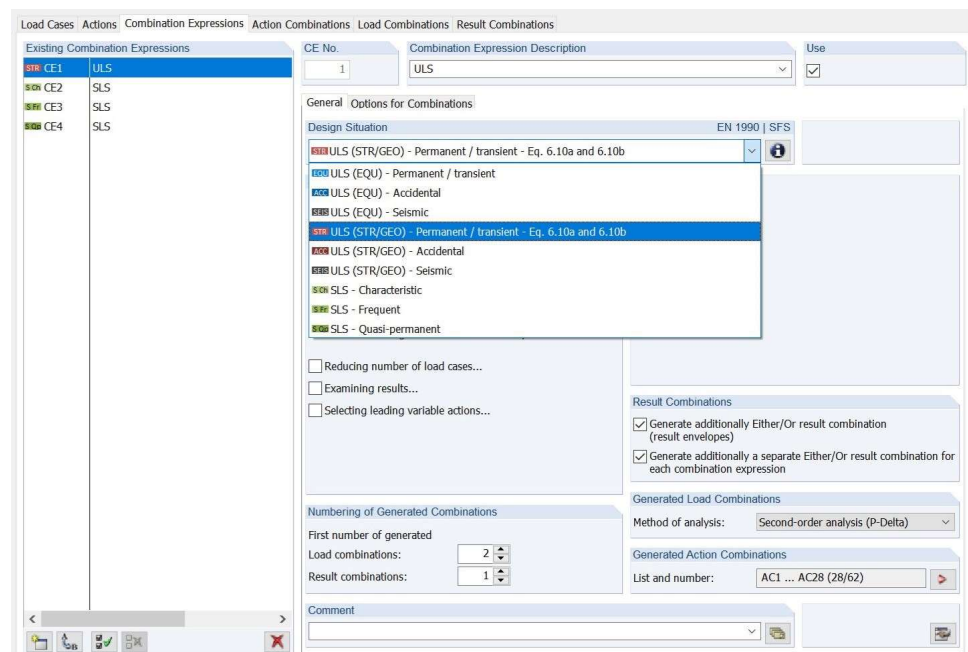


Figure 15. Design situations

EN 1990 | SFS
ULS (STR/GEO) - Permanent / transient - Eq. 6.10a and 6.10b

$$\sum_{j \geq 1} \gamma_{G,j} G_{k,j} + \gamma_P P$$

$$\sum_{j \geq 1} \xi_j \gamma_{G,j} G_{k,j} + \gamma_P P + \gamma_{Q,1} Q_{k,1} + \sum_{i > 1} \gamma_{Q,i} \psi_{0,i} Q_{k,i}$$

$\gamma_{G,j}$: Partial factor for permanent actions
 $G_{k,j}$: Permanent actions
 γ_P : Partial factor for prestress actions
 P : Prestressing action
 $\gamma_{Q,1}$: Partial factor for leading variable action
 $Q_{k,1}$: Leading variable action
 $\gamma_{Q,i}$: Partial factor for non-leading variable actions
 $\psi_{0,i}$: Coefficient for combination value
 $Q_{k,i}$: Other variable actions
 ξ_j : Reduction coefficient

Figure 16. Information about design situation

7.3 Loads and Load combinations

Dlubal RFEM provides essential tools for generating wind, snow, surface, and member loads and automatically calculates the self-weights of all elements and total mass of the building (see Figure 17). Generated load cases are manually assigned to an action category, which includes partial safety factors and combination coefficients (Figure 18). The loads are then applied to desired elements and are visible in a Load case table as seen in Figure 19 on the next page.

The screenshot shows the 'Mass Cases' dialog box in the software. The 'Existing Mass Cases' list on the left contains 'MC1'. The 'General' tab is selected, showing the following configuration:

- MC No.:** 1
- Mass Case Description:** MC1
- Mass Case Type:** Permanent
- Masses:**
 - From self-weight of structure
 - From force components of:
 - Load case: I.C1 - FloorDL
 - Load combination: CO1 - Load Combination 1
- Manually define additional masses at:**
 - Nodes
 - Lines
 - Members
 - Surfaces
- Sum of Masses:**
 - Self-weight:** 7201726.08 [kg]
 - Components of LC/CO:** [Empty field]
 - Additional masses at:**
 - Nodes:** [Empty field] [kg]
 - Lines:** [Empty field] [kg]
 - Members:** [Empty field] [kg]
 - Surfaces:** [Empty field] [kg]
 - Total mass:** 7201726.08 [kg]
 - Center of total mass Coordinates X, Y, Z:** [?] [m]

A 'Calculate Masses' button is located at the bottom right of the dialog.

Figure 17. Sum of masses

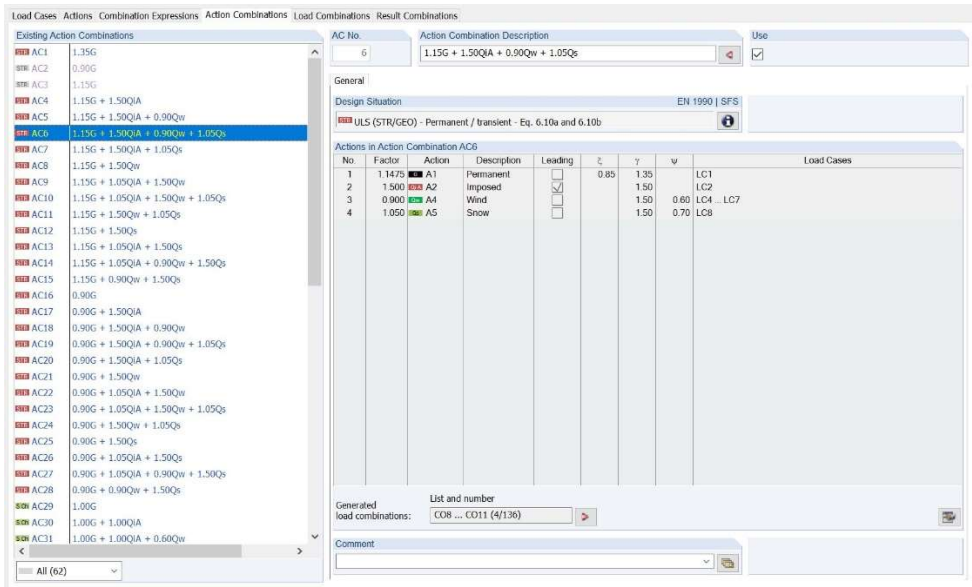


Figure 18. Action combinations

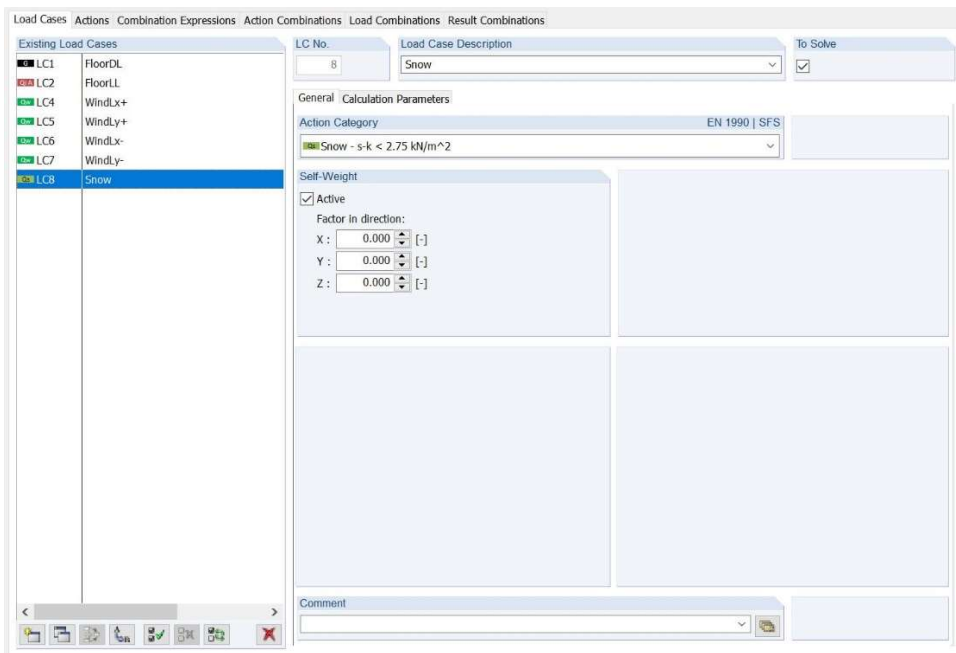


Figure 19. Load cases

After entering type and magnitude of the load, it is assigned to nodes, lines, members or surfaces as seen in Figure 20. The values of permanent loads and properties of elements are set with estimation to allow a margin of error, which will allow future alteration of the model to exceed its parameters. Acting loads on the structure can be seen in Appendix 3.

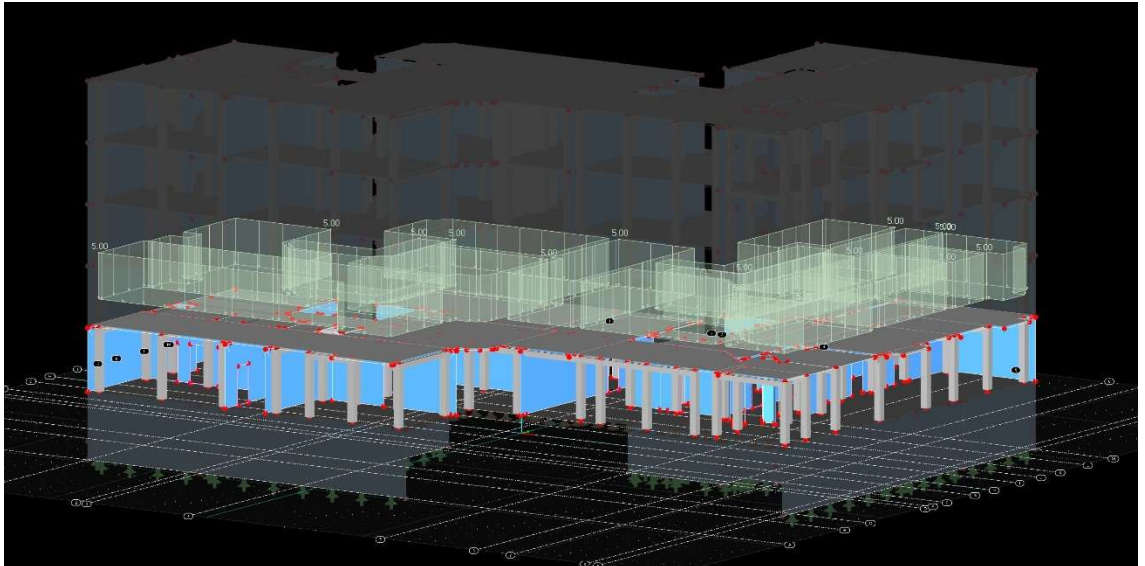


Figure 20. First floor Dead Load

Generation of snow and wind loads in RFEM is done with no additional modules. When the required parameters are filled, the program generates specific load automatically. Required parameters include wind and snow load zones, topography, terrain and roof geometry (Figure 21). Additional snow loads such as snow drifts are considered, too (see Figure 22).

Velocity Pressure According to Standard: EN 1991-1-4 National Annex: SFS Wind zone: I Terrain category: Category IV <input checked="" type="checkbox"/> Lack of correlation acc. to 7.2.2(3)		Structure height h : 16.770 [m]	Vertical Walls
Base Geometry Node I: 1284 Node J: 1430 Node K: 1329 Node L: 1264		Fundamental wind velocity $v_{b,0}$: 21.00 [m/s]	
Roof Type and Geometry Type: <input type="radio"/> Flat/monopitch roof... <input checked="" type="radio"/> Duopitch roof			
Load Cases to Generate <input checked="" type="checkbox"/> LC w: LCS		Set Wind on Side <input type="radio"/> A - B - C <input type="radio"/> D - E - F <input type="radio"/> C - D <input type="radio"/> F - A	
Internal Pressure <input type="checkbox"/> Activate internal pressure <input type="checkbox"/> Consider only increasing loads C_{pi} : [-]			
Create Load Type <input checked="" type="radio"/> Member loads <input type="radio"/> Surface loads		Load Distribution Type <input type="radio"/> Axes of angles <input checked="" type="radio"/> Combined <input type="radio"/> Constant	
Remove Influence from Single members:		Members parallel to member:	
		Generate Wind Loads on Members No. 121,141,169,172,173,232,236,251,284,380,399,421,583,601	

Figure 21. Wind loads

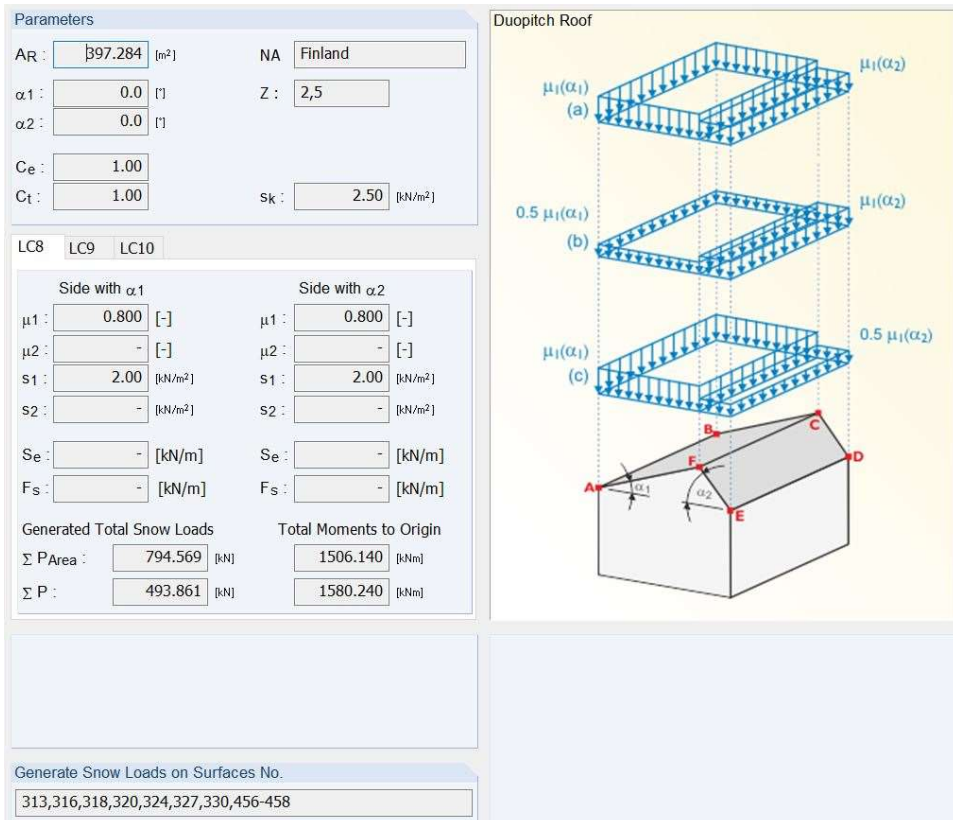


Figure 22. Snow drift loads

Taking the partial safety factor into account, the program combines assigned load cases into one big load case and subsequently calculates it.

7.4 Finite element analysis

Analysing a phenomenon with FEM is referred to as FEA which breaks down the structure into many small substructures, finite elements. The generated structure which is identical to the original structure is referred to as the Finite Element (FE) mesh Figure 23.

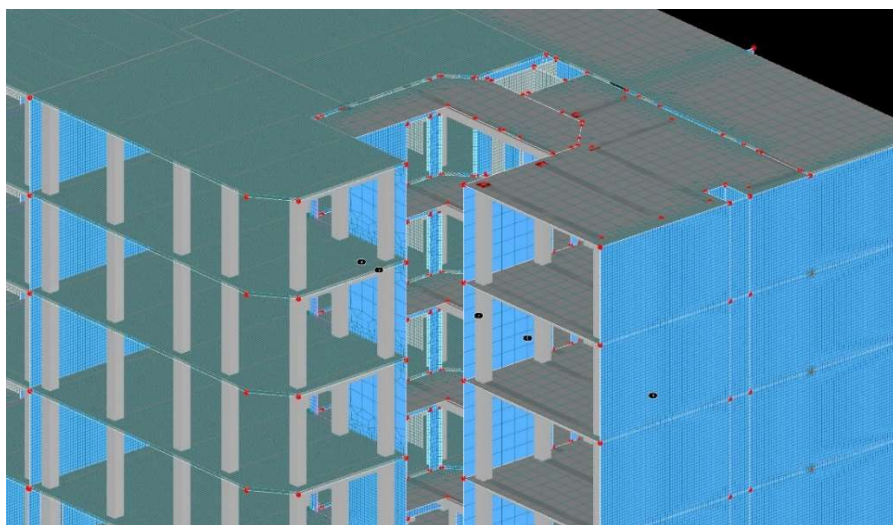


Figure 23. Finite Element mesh

The size or the number of finite elements affects the precision of results. The smaller the elements are set; the closer the analysis gets to the ideal result. The proportional relation also affects the duration of the analysis.

8 STRUCTURAL ANALYSIS RESULTS

8.1 Result summary

While performing FE analysis it is important to remember that it is an approximation. The results should always be checked and clarified with engineering expertise. When the analysis is complete the results are highlighted in different colours, scaling values of internal forces and deformations from minimum to extreme, allowing visual result evaluation (see Figure 24).

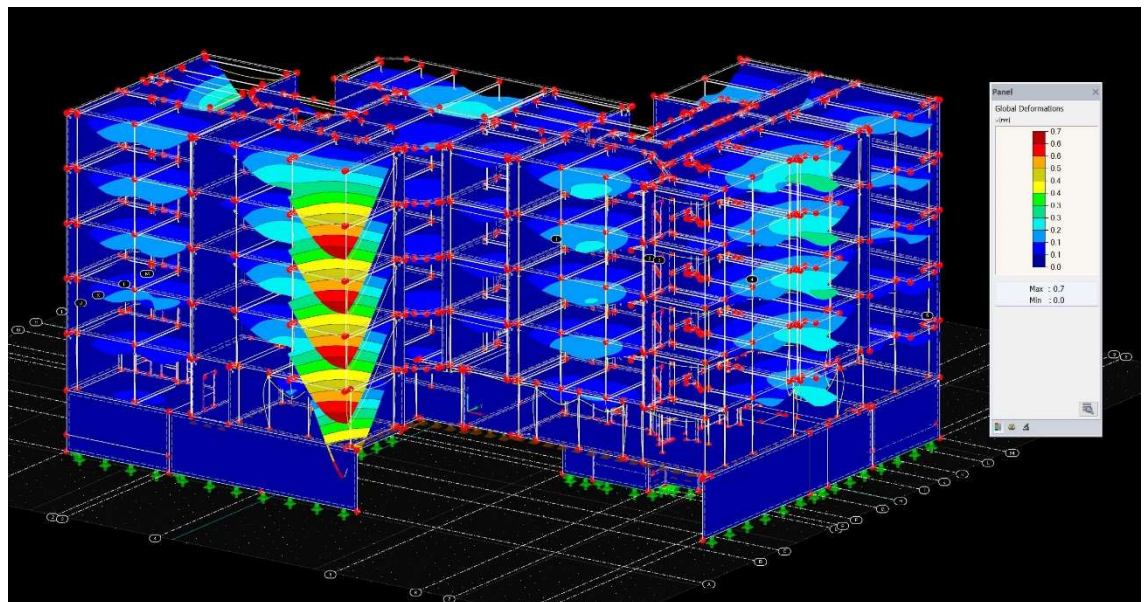


Figure 24. Visual result evaluation

RFEM provides a summary of the results sorted by load cases and combinations as shown in Table 2 on the next page. The overview shows maximum displacements and rotations related to the global axes, as well as the total largest displacement. The deviation in each direction should be less than 1%, signifying that no numerical problems related to stiffness difference or structure stability have occurred. The summary is completed by selected calculation parameters of the analysis core and specifications of calculations. Different views of analysed structure can be seen in Appendix 4.

Table 2. Results – Summary

4.0 Results - Summary			
A	B	C	D
Description	Value	Unit	Comment
LC1 - FloorDL			
Sum of loads in X	0.00	kN	
Sum of support forces in X	0.00	kN	
Sum of loads in Y	0.00	kN	
Sum of support forces in Y	0.00	kN	
Sum of loads in Z	-93996.00	kN	
Sum of support forces in Z	-93996.00	kN	Deviation: 0.00 %
Resultant of reactions about X	11563.100	kNm	At center of gravity of model (X: 6.689, Y: -4.905, Z: 10.285 m)
Resultant of reactions about Y	10801.400	kNm	At center of gravity of model
Resultant of reactions about Z	-0.010	kNm	At center of gravity of model
Maximum displacement in X-direction	-0.4	mm	Member No. 589, x: 2.070 m
Maximum displacement in Y-direction	0.6	mm	Member No. 703, x: 2.415 m
Maximum displacement in Z-direction	-76.7	mm	Member No. 60, x: 2.090 m
Maximum vectorial displacement	76.7	mm	Member No. 60, x: 2.090 m
Maximum rotation about X-axis	13.4	mrاد	Member No. 60, x: 2.090 m
Maximum rotation about Y-axis	-5.6	mrاد	Member No. 955, x: 0.000 m
Maximum rotation about Z-axis	0.3	mrاد	Member No. 155, x: 0.000 m
Method of analysis	Linear		Geometrically Linear Analysis
Reduction of stiffness	<input type="checkbox"/>		
Number of load increments	1		
Number of iterations	1		
Maximum value of element of stiffness matrix on dia	2.198E+12		
Minimum value of element of stiffness matrix on diag	4.748E+04		
Infinity Norm	5.295E+12		
Incrementally increasing loading	<input type="checkbox"/>		
LC2 - FloorLL			
Sum of loads in X	0.00	kN	
Sum of support forces in X	0.00	kN	
Sum of loads in Y	0.00	kN	
Sum of support forces in Y	0.00	kN	
Sum of loads in Z	-6443.29	kN	
Sum of support forces in Z	-6443.29	kN	Deviation: 0.00 %
Resultant of reactions about X	2858.780	kNm	At center of gravity of model (X: 6.689, Y: -4.905, Z: 10.285 m)
Resultant of reactions about Y	3523.990	kNm	At center of gravity of model
Resultant of reactions about Z	0.000	kNm	At center of gravity of model
Maximum displacement in X-direction	0.0	mm	Member No. 589, x: 2.070 m
Maximum displacement in Y-direction	0.1	mm	Member No. 703, x: 2.415 m
Maximum displacement in Z-direction	-0.7	mm	FE Mesh Node No. 465967 (X: 4.386, Y: -19.570, Z: 20.440 m)
Maximum vectorial displacement	0.7	mm	FE Mesh Node No. 465967 (X: 4.386, Y: -19.570, Z: 20.440 m)
Maximum rotation about X-axis	0.2	mrاد	FE Mesh Node No. 637231 (X: 4.868, Y: -16.195, Z: 7.120 m)
Maximum rotation about Y-axis	0.3	mrاد	FE Mesh Node No. 637275 (X: 3.681, Y: -19.570, Z: 7.120 m)
Maximum rotation about Z-axis	0.0	mrاد	Member No. 224, x: 0.000 m
Method of analysis	Linear		Geometrically Linear Analysis
Reduction of stiffness	<input type="checkbox"/>		
Number of load increments	1		
Number of iterations	1		
Maximum value of element of stiffness matrix on dia	2.198E+12		
Minimum value of element of stiffness matrix on diag	4.748E+04		
Infinity Norm	5.295E+12		
Incrementally increasing loading	<input type="checkbox"/>		

8.2 Deformations

The display of surface and member displacements and rotations are presented both graphically and numerically. Deformations of elements can be determined for members, surfaces and solids and are related either to the local coordinate system of selected element or to global coordinate system. Accessed through Result navigator, deformations have the following meanings: U for displacement and ϕ for rotation (see Table 3). Locally selected deformations list deformations of each member separately and follows as start and end of the member, division points and extreme values, where surface deformations are defined by FE meshes grid points which represent its characteristics, as seen in Figure 25.

Table 3. Global deformations

4.5 Members - Global Deformations										
Member No.	Node No.	Location x [m]	C	Displacements [mm]			Rotations [mrad]			Corresponding Load Cases
				ux	uy	uz	φx	φy	φz	
59	2540	0.000	max	0.0	0.0	-6.3	1.3	2.0	0.0	
			min	0.0	0.0	-6.3	1.3	2.0	0.0	
	115	2.090	max	0.0	0.0	-9.3	1.5	2.0	0.0	
			min	0.0	0.0	-9.3	1.5	2.0	0.0	
	2540	0.000	Max ux	0.0	0.0	-6.3	1.3	2.0	0.0	CO 98
	115	2.090	Min ux	0.0	0.0	-9.3	1.5	2.0	0.0	CO 89
	2540	0.000	Max uy	0.0	0.0	-6.3	1.3	2.0	0.0	CO 89
	2540	0.000	Min uy	0.0	0.0	-6.3	1.3	2.0	0.0	CO 98
	2540	0.000	Max uz	0.0	0.0	-6.3	1.3	2.0	0.0	CO 87
	115	2.090	Min uz	0.0	0.0	-9.3	1.5	2.0	0.0	CO 85
	115	2.090	Max φx	0.0	0.0	-9.3	1.5	2.0	0.0	CO 84
	2540	0.000	Min φx	0.0	0.0	-6.3	1.3	2.0	0.0	CO 90
	2540	0.000	Max φy	0.0	0.0	-6.3	1.3	2.0	0.0	CO 90
	2540	0.000	Min φy	0.0	0.0	-6.3	1.3	2.0	0.0	CO 84
	2540	0.000	Max φz	0.0	0.0	-6.3	1.3	2.0	0.0	CO 90
	2540	0.000	Min φz	0.0	0.0	-6.3	1.3	2.0	0.0	CO 97
60	2529	0.000	max	0.0	0.0	-48.8	13.2	-4.9	0.0	
			min	0.0	0.0	-48.8	13.2	-4.9	0.0	
	114	2.090	max	0.0	0.0	-76.7	13.4	-4.9	0.0	
			min	0.0	0.0	-76.7	13.4	-4.9	0.0	
	114	2.090	Max ux	0.0	0.0	-76.7	13.4	-4.9	0.0	CO 94
	2529	0.000	Min ux	0.0	0.0	-48.8	13.2	4.9	0.0	CO 97
	2529	0.000	Max uy	0.0	0.0	-48.8	13.2	-4.9	0.0	CO 97
	2529	0.000	Min uy	0.0	0.0	-48.8	13.2	-4.9	0.0	CO 90
	2529	0.000	Max uz	0.0	0.0	-48.8	13.2	-4.9	0.0	CO 89
	114	2.090	Min uz	0.0	0.0	-76.7	13.4	-4.9	0.0	CO 85
	114	2.090	Max φx	0.0	0.0	-76.7	13.4	-4.9	0.0	CO 90
	2529	0.000	Min φx	0.0	0.0	-48.8	13.2	-4.9	0.0	CO 84
	2529	0.000	Max φy	0.0	0.0	-48.8	13.2	-4.9	0.0	CO 90
	2529	0.000	Min φy	0.0	0.0	-48.8	13.2	-4.9	0.0	CO 97
	2529	0.000	Max φz	0.0	0.0	-48.8	13.2	-4.9	0.0	CO 85
	2529	0.000	Min φz	0.0	0.0	-48.8	13.2	-4.9	0.0	CO 89
61	2661	0.000	max	0.0	0.0	-6.3	1.3	2.0	0.0	
			min	0.0	0.0	-6.3	1.3	2.0	0.0	
	117	2.090	max	0.0	0.0	-9.3	1.5	2.0	0.0	
			min	-0.1	0.0	-9.4	1.5	2.0	0.0	
	2661	0.000	Max ux	0.0	0.0	-6.3	1.3	2.0	0.0	CO 98
	117	2.090	Min ux	-0.1	0.0	-9.3	1.5	2.0	0.0	CO 89
	2661	0.000	Max uy	0.0	0.0	-6.3	1.3	2.0	0.0	CO 89
	2661	0.000	Min uy	0.0	0.0	-6.3	1.3	2.0	0.0	CO 85
	2661	0.000	Max uz	0.0	0.0	-6.3	1.3	2.0	0.0	CO 90
	117	2.090	Min uz	0.0	0.0	-9.4	1.5	2.0	0.0	CO 84
	117	2.090	Max φx	0.0	0.0	-9.4	1.5	2.0	0.0	CO 84
	2661	0.000	Min φx	0.0	0.0	-6.3	1.3	2.0	0.0	CO 90
	2661	0.000	Max φy	0.0	0.0	-6.3	1.3	2.0	0.0	CO 90
	2661	0.000	Min φy	0.0	0.0	-6.3	1.3	2.0	0.0	CO 84
	2661	0.000	Max φz	0.0	0.0	-6.3	1.3	2.0	0.0	CO 90
	2661	0.000	Min φz	0.0	0.0	-6.3	1.3	2.0	0.0	CO 84

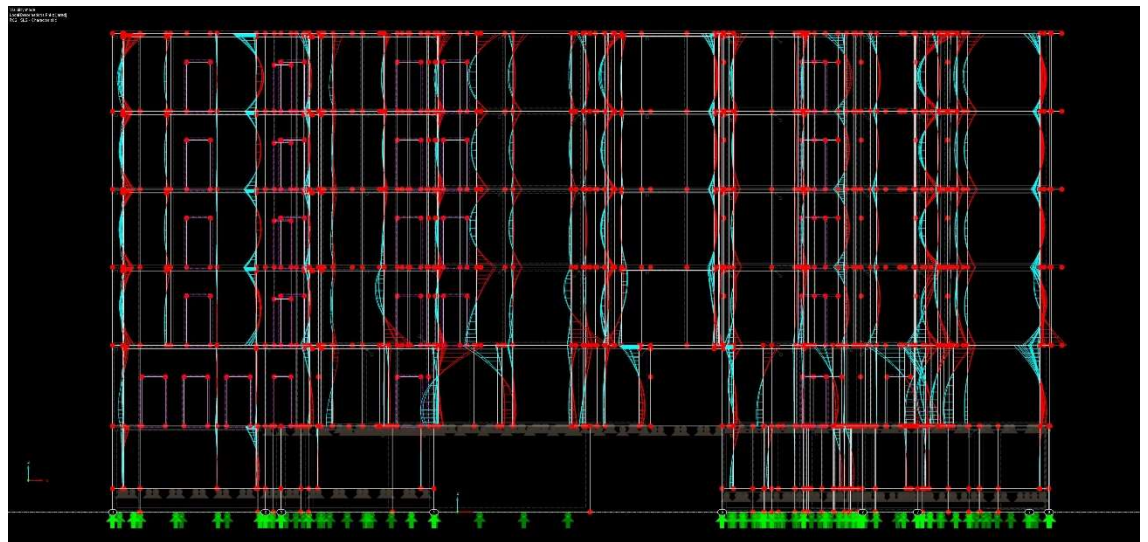


Figure 25. Local deformations

8.3 Support reactions

It is also possible to evaluate support reactions graphically or in tables. Support reactions are listed in lines F and are related to the start node of the line. The surface FE mesh is not relevant to the line support forces.

Presented in results, supporting reactions are generated for the individual load case and combination.

8.4 Internal forces

Internal forces are axial forces, shear forces and moments acting within a body. Presented both graphically and numerically internal forces are sorted by surfaces or members. The difference between surface internal forces and member internal forces is that a member moment M rotates about the local axis of a member, whereas surface moment acts in the direction of local surface axis.

8.5 Structural components design

In addition to program's ability to determine deformations, support reactions and internal forces, RFEM provides several add-on modules, which can perform structural design of elements consisting of different materials and according to specific standards.

The RF-CONCRETE add-on module is used to determine cracked sections in reinforced concrete structures and to design structural components. The module consists of two parts: RF-CONCRETE Members and RF-CONCRETE Surfaces.

8.5.1 RF-CONCRETE

Members designed through RF-Concrete are imported directly from RFEM. Loads, load combinations and result combinations are then assigned to determine internal forces of selected elements. The library covers the property of materials of the relevant standard and can be altered if necessary (Figure 26).

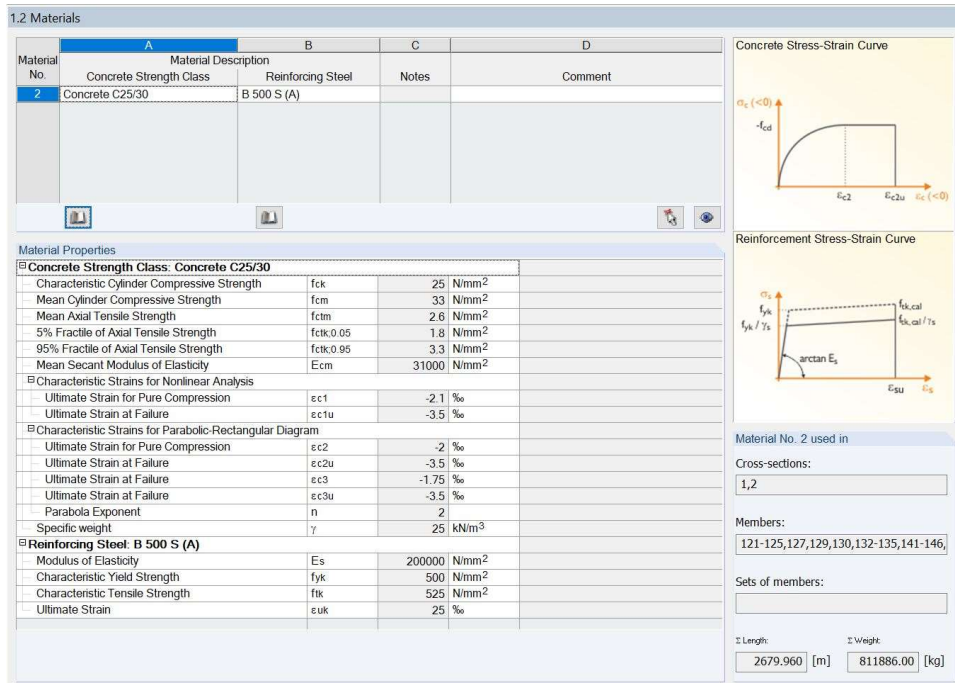


Figure 26. Material Properties

The RF-Concrete Member add-on is used to design reinforcement for beams, columns and other elements, where RF-Concrete Surfaces is used to design reinforcement for surface. Prior to design calculation, construction properties are set to determine the required longitudinal and shear reinforcement. During this project possible diameters of longitudinal reinforcement are set from 16 to 28 mm and from 8 to 10 mm for stirrups. The results of design are then presented both in tables and graphically, visualizing the most suitable solution (see Figure 27). Structural designs generated by RF-Concrete can be seen in Appendix 5.

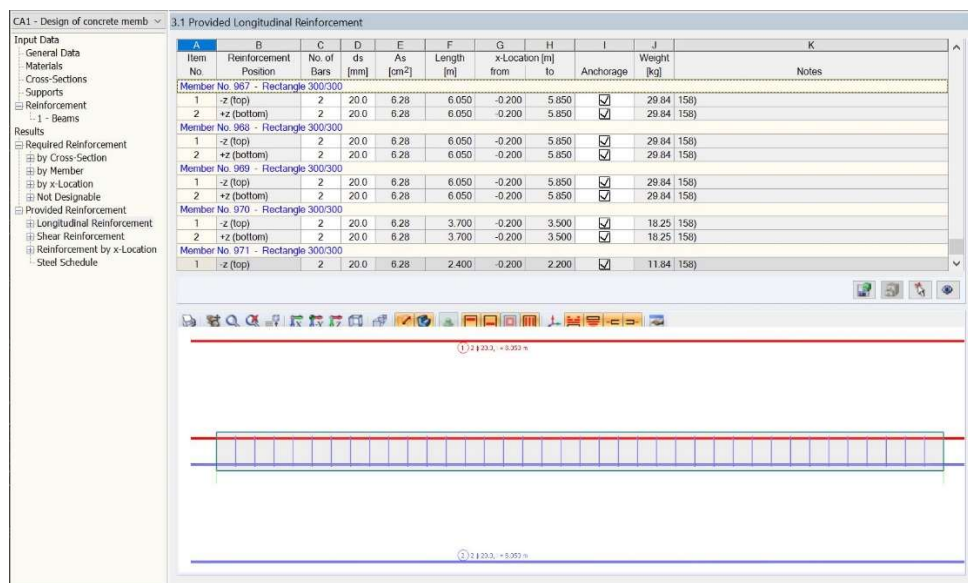


Figure 27. Provided reinforcement

8.6 Updating existing BIM

To complete building information modelling, structural analysis is exported from Dlubal RFEM to Autodesk Revit. The program's bidirectional data exchange makes it possible to create new elements and adjust modifications in both directions. Adjustable reference lines allow load bearing structural parts to merge automatically in a single model. If changes are made to cross-sections or materials, the data is then updated in both programs. New structural components are added, and modified designs are removed as seen in Figure 28.

The results of structural analysis such as internal forces, deformations and defined reinforcement groups can also be imported to Revit. A corresponding view as well as Result Explorer toolkit is generated automatically and is available after import is complete. Manual adjustments of the analytical model are always possible in case of occurred inaccuracies. Exported elements of the analytical model can be found in Appendix 6.

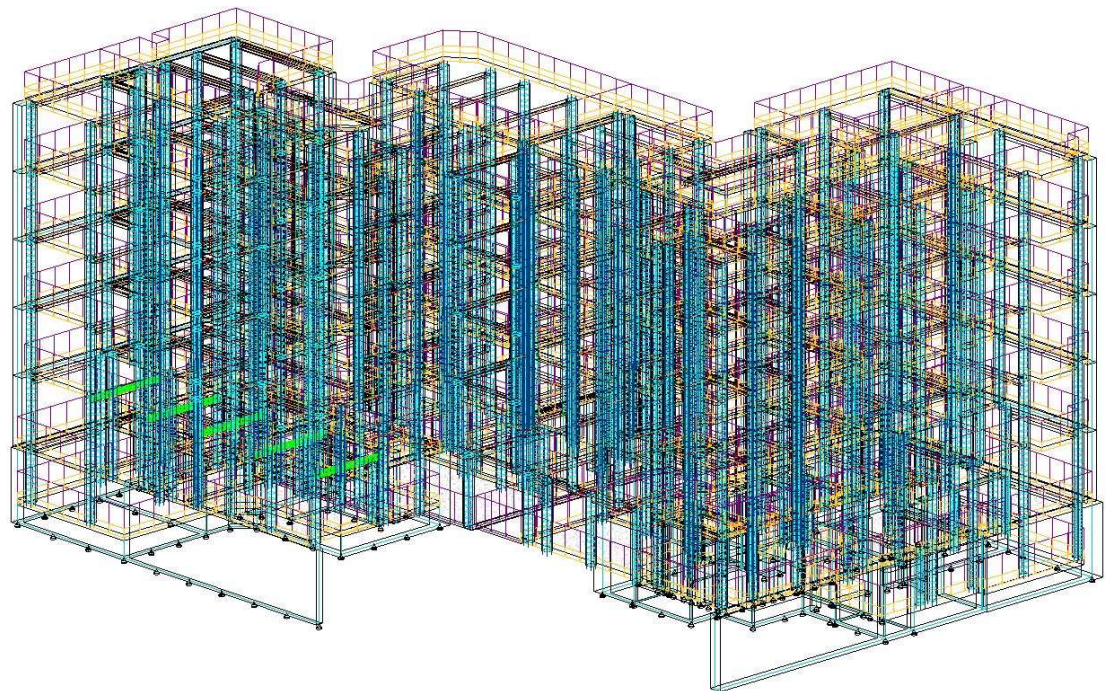


Figure 28. Exported reinforcement

9 CONCLUSION

The availability of information online and provision of free student versions of the programs by Autodesk and Dlubal software, makes it possible for those who are interested, to learn operating the programs efficiently. This makes the studying process relatively fast and easy.

Revit and RFEM mostly circulate amongst large construction companies, but most, if not all the features and libraries are accessible in student and trial versions of programs, for separate users to experience the software prior to its purchase. Programs used during this project not only provided results for structural analysis and combination of them in a BIM, but also gave insights into learning to use Revit and RFEM and laid the basis for a potential future renovation.

The presence of link between two programs allowed the project to expand its possibilities. The program's tools effectively assisted in conflict determination and its further elimination, making the next step of the process possible when certain condition is fulfilled. Although there is always room for improvement, the approach taken during designing stage of the project is represented through a flowchart seen in Figure 29, compiled for better understanding of the workflow.

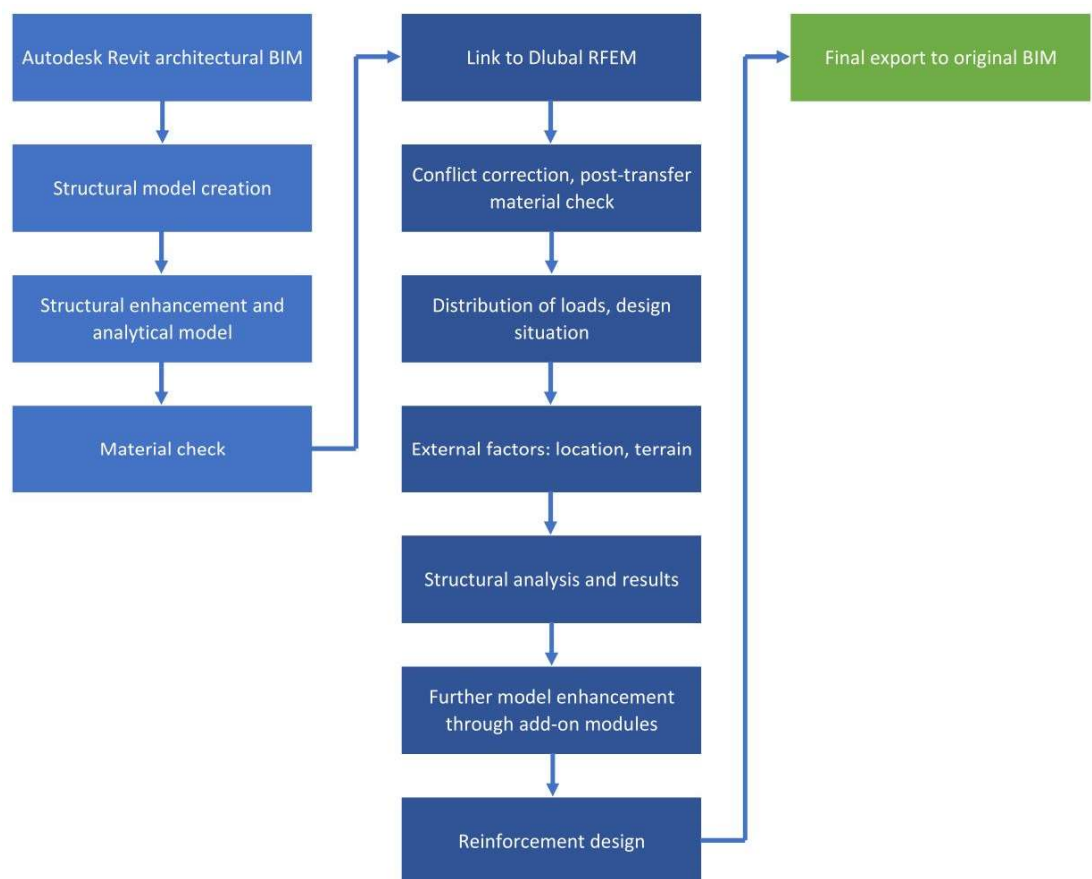


Figure 29. Flowchart of the project

REFERENCES

- Autodesk. (2019). Office locations. Retrieved 5 February 2019 from <https://www.autodesk.com/company/contact-us/office-locations>
- Bergin M. (2012). A brief history of BIM. *Style of design - Architecture* 7 December 2012. Retrieved 15 January 2019 from <https://web.archive.org/web/20140302064642/http://www.styleofdesign.com/architecture/a-brief-history-of-bim-michael-s-bergin/>
- BIM Wiki. (2019). Parametric modelling. Retrieved 13 February 2019 from https://www.designingbuildings.co.uk/wiki/Parametric_modelling
- Cherkaoui H. (2016). What is BIM? Blog publication 1 December 2016. Retrieved 20 January 2019 from <https://www.letsbuild.com/blog/what-is-bim-what-are-its-benefits-to-the-construction-industry>
- Dlubal software. (2019). Company in facts. Retrieved 15 March 2019 from <https://www.dlubal.com/en/company/about-us/history-and-facts>
- Finite element method (FEM). (2019). Software for Finite Element Analysis (FEA). Retrieved 20 March 2019 from <https://www.dlubal.com/en/solutions/application-areas/finite-element-analysis-fea-software>
- Funding Universe. (n.d.) Autodesk, Inc. History. Retrieved 1 February 2019 from <http://www.fundinguniverse.com/company-histories/autodesk-inc-history/>
- Helsinki kuvia. (n.d.). Fleminginkatu 10. Franzeninkatu 23. Retrieved 18 January 2019 from <https://www.helsinkikuvia.fi/collection/694/?rid=11703>
- Herkaoui H. (2017). A history of BIM. Blog publication 22 March 2017. Retrieved 25 January 2019 from <https://www.letsbuild.com/blog/a-history-of-bim>
- Rakennuslupa. (2016). As Oy Helsingin Franzeninaukio. Retrieved 3 March 2017 from <https://www.hel.fi/static/rakvv/VIPA/LYV/2016/55-LYVPTK.pdf>
- RFEM. (2019). FEM Structural Analysis Software. Retrieved 10 April 2019 from <https://www.dlubal.com/en/products/rfem-fea-software/what-is-rfem>

Technopedia. (2012). Object-oriented modelling. Retrieved 20 April 2019 from <https://www.techopedia.com/definition/28584/object-oriented-modeling-oom>

Appendix 1/1
Visible crack on the old concrete slab



Appendix 1/2
Cracks formed around old ventilation shaft



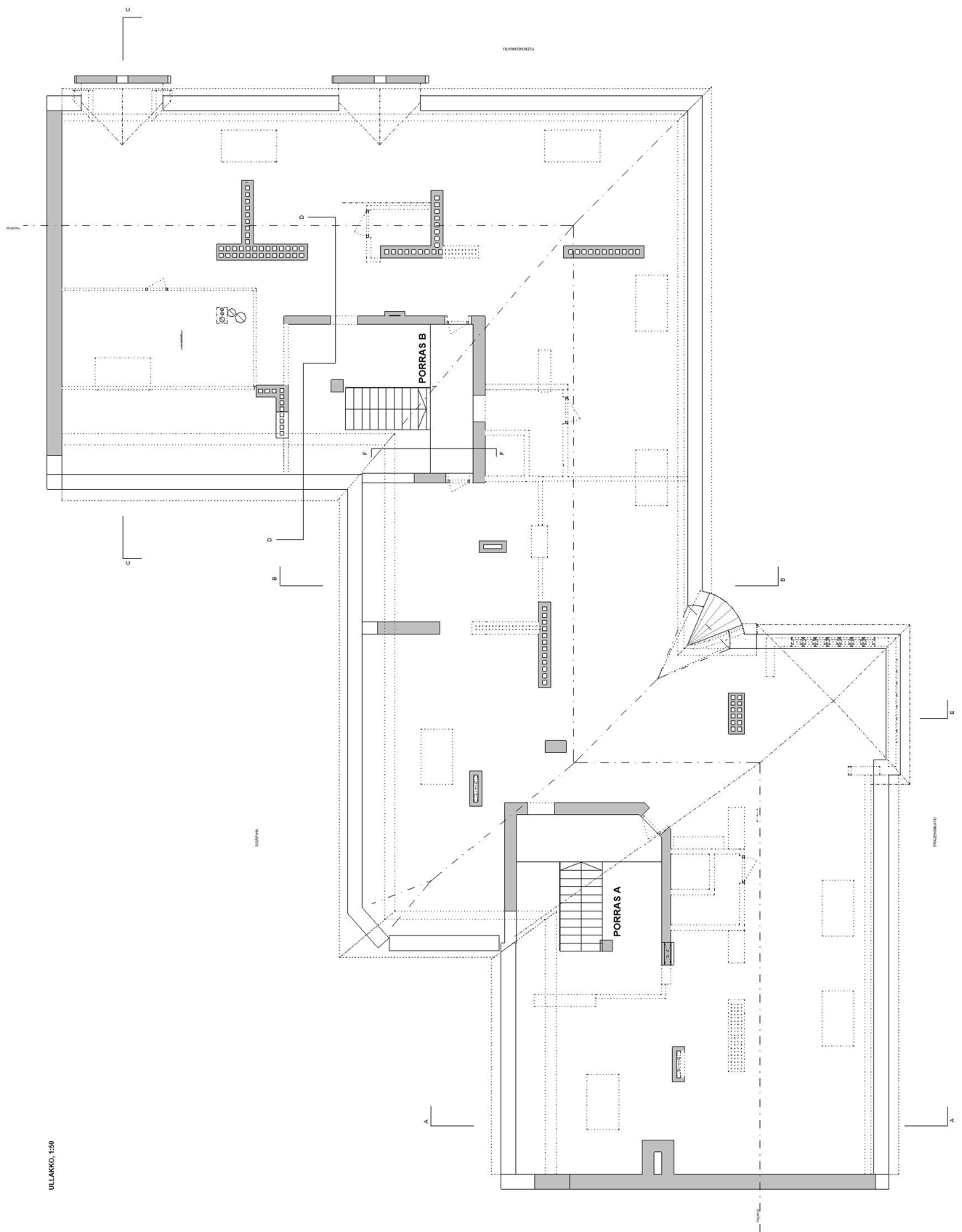
Appendix 1/3
Reinforcement of the old the damaged concrete slab



Appendix 1/4
Reinforcement of deteriorating concrete slab

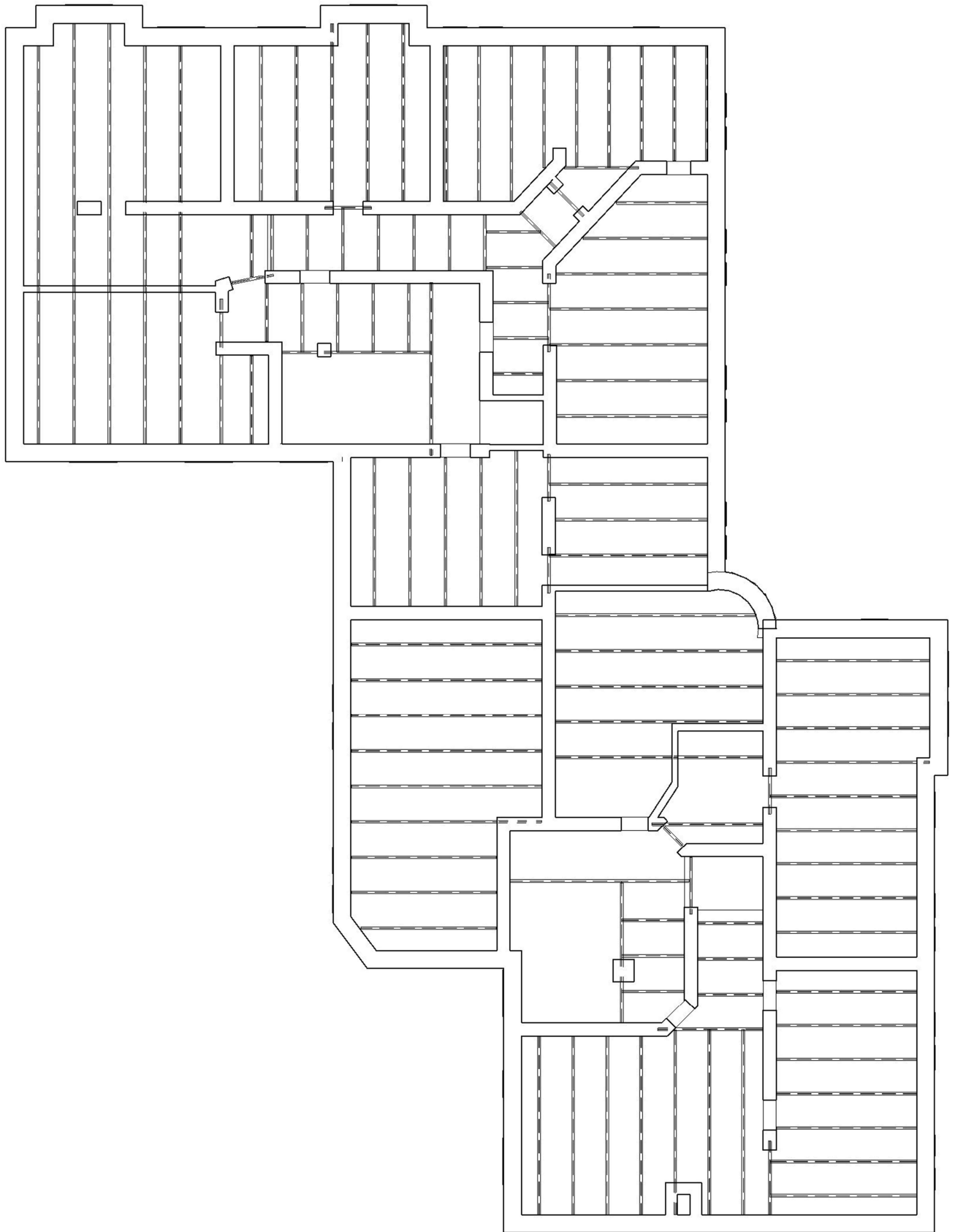


Appendix 2/1
Autodesk Revit generated model and drawings
Demolition drawing of attic structures



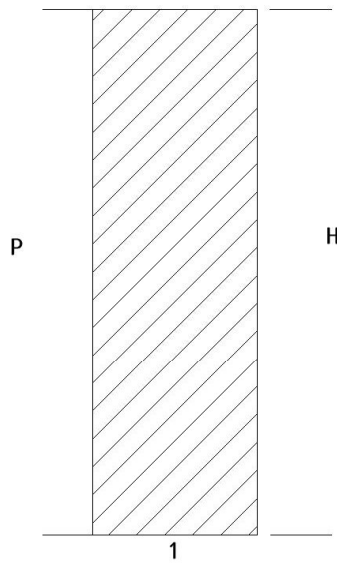
UILLANKKO, 1:50

Appendix 2/2
Layout of masonry walls and old INP 200-profiles



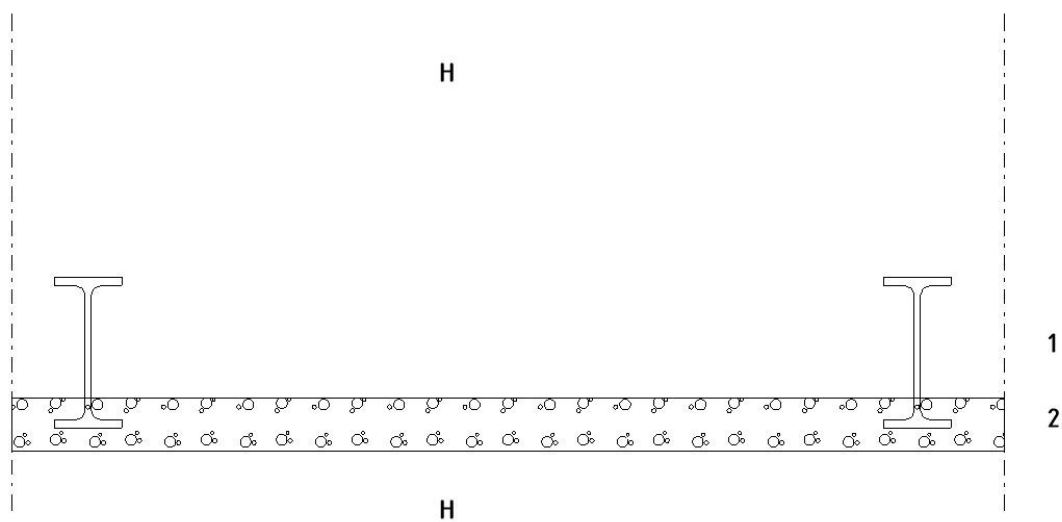
Appendix 2/3
Masonry wall and intermediate floor section view

1:10



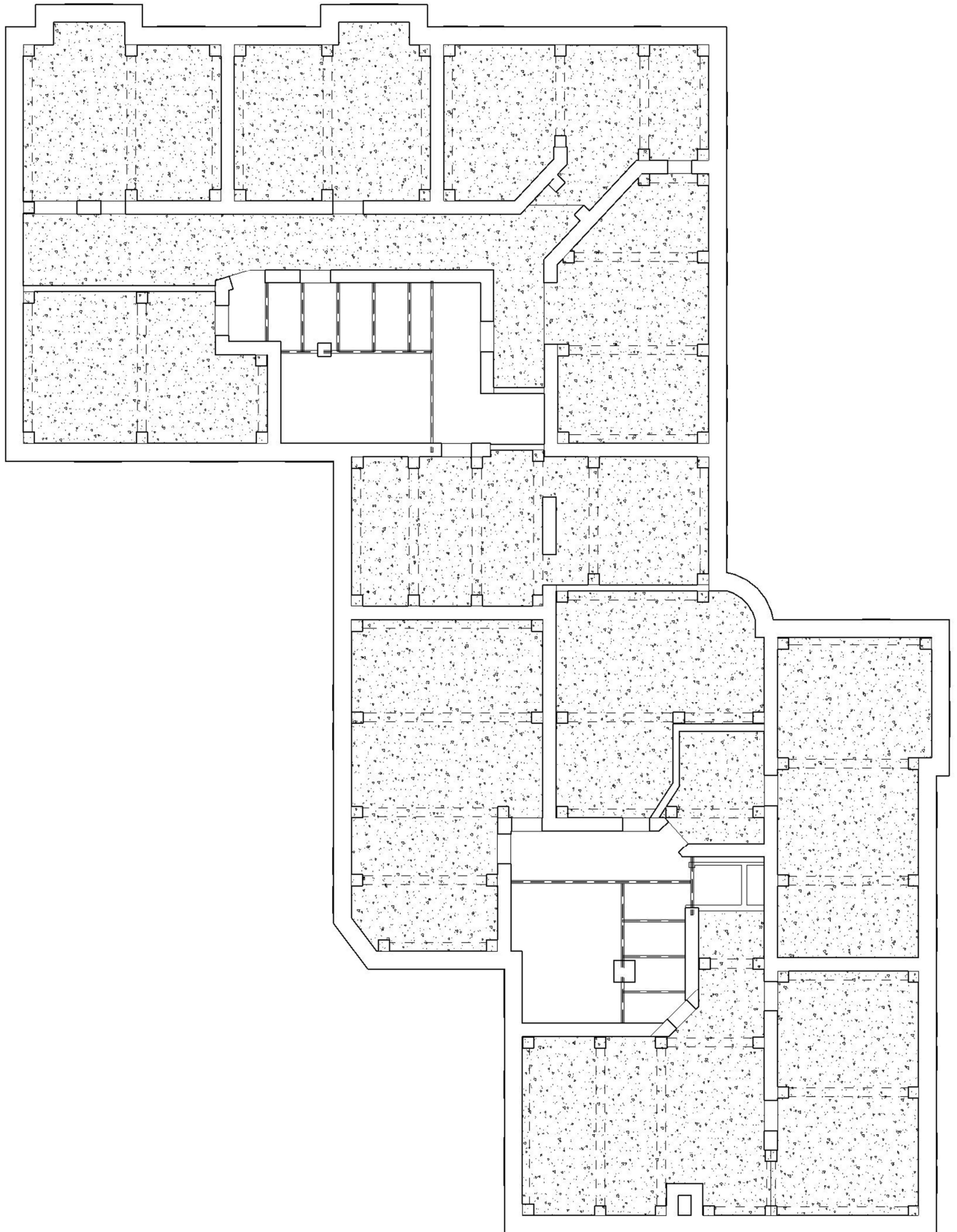
1 OLEMASSA OLEVA KIVIRAKENTEINEN SEINÄ

1:100

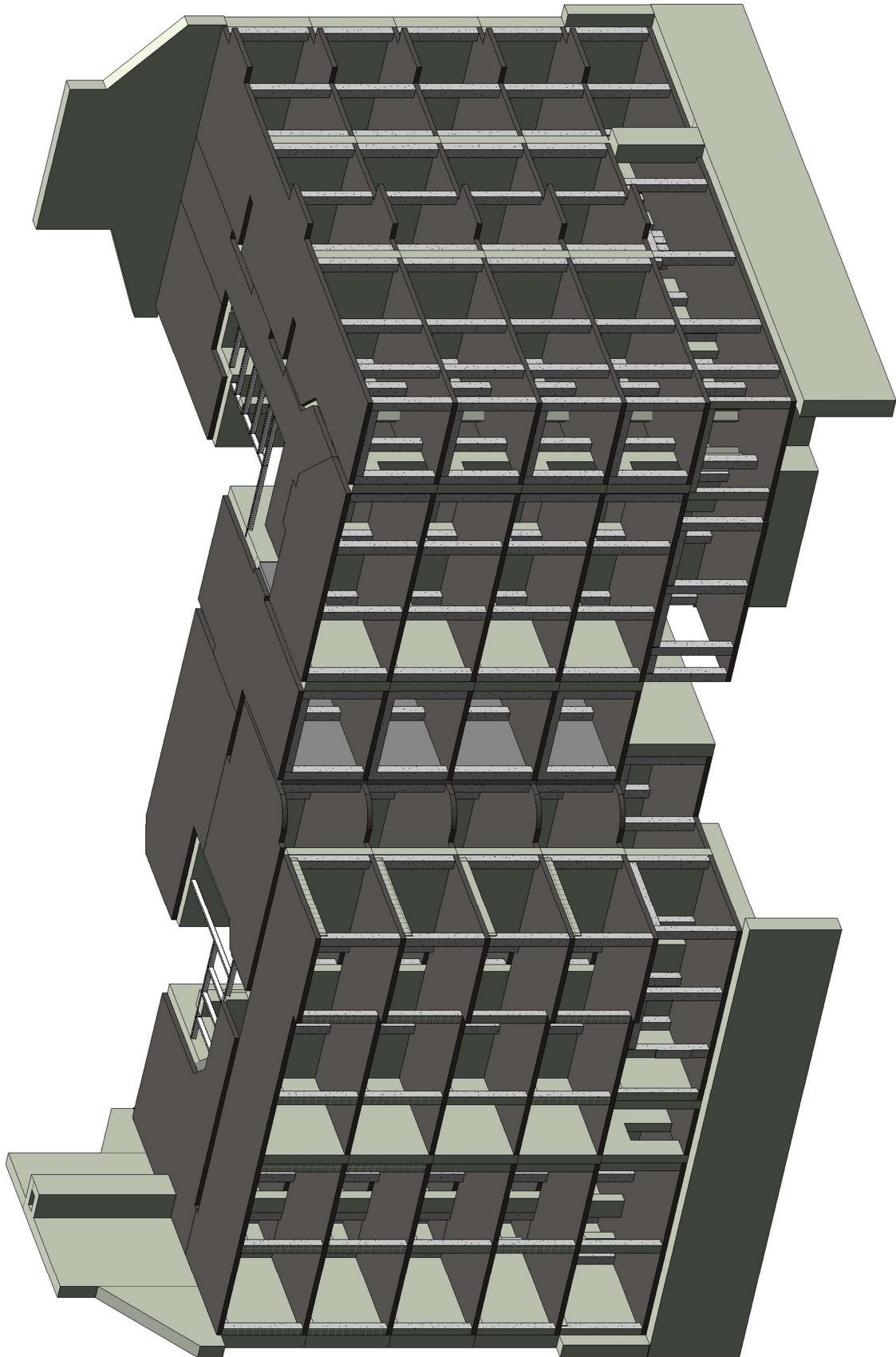


1 OLEMASSA OLEVAT I-PALKIT
2 BETONIALAATTA 150 MM

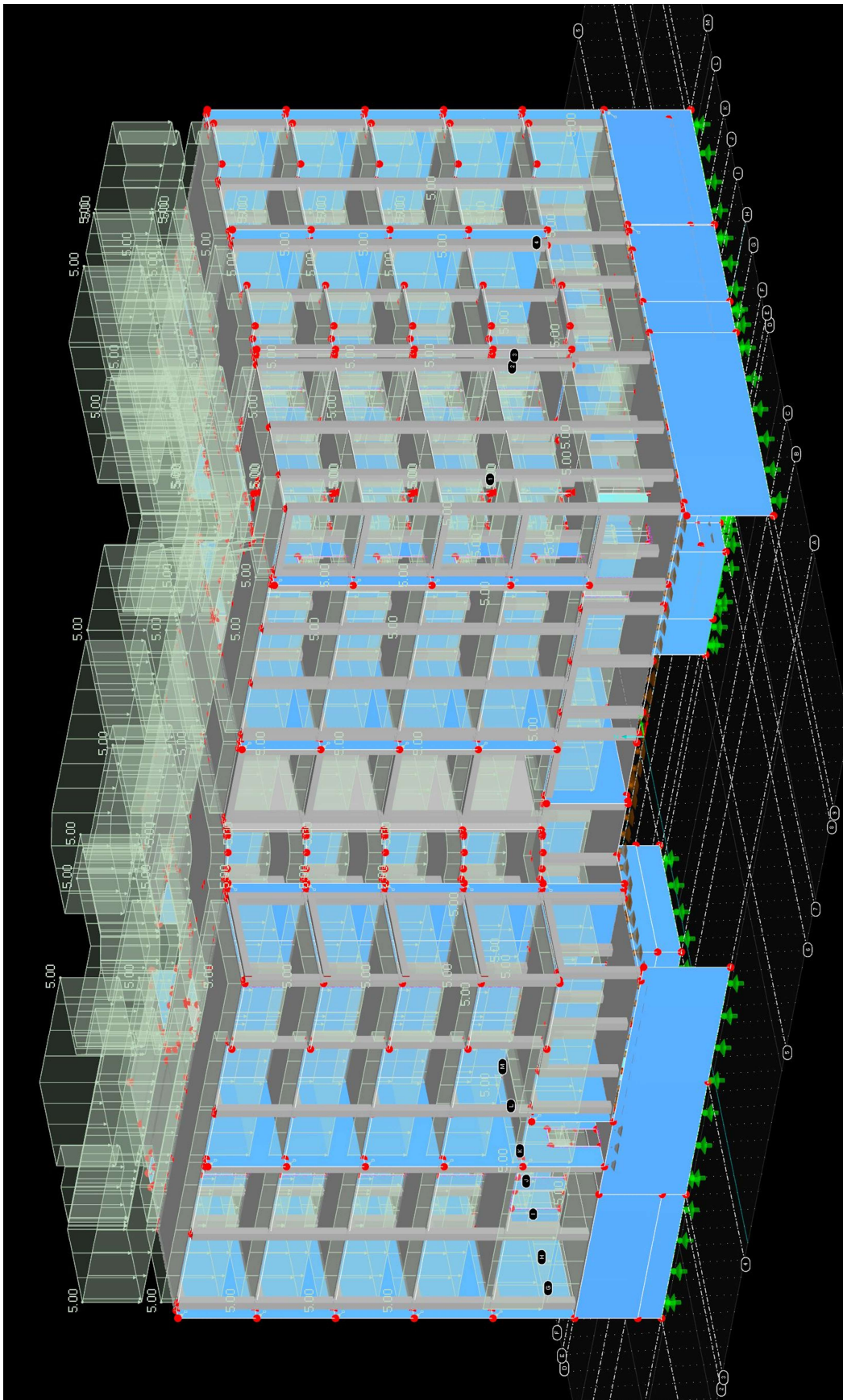
Appendix 2/4
Layout of the new structural frame 2-5 floor



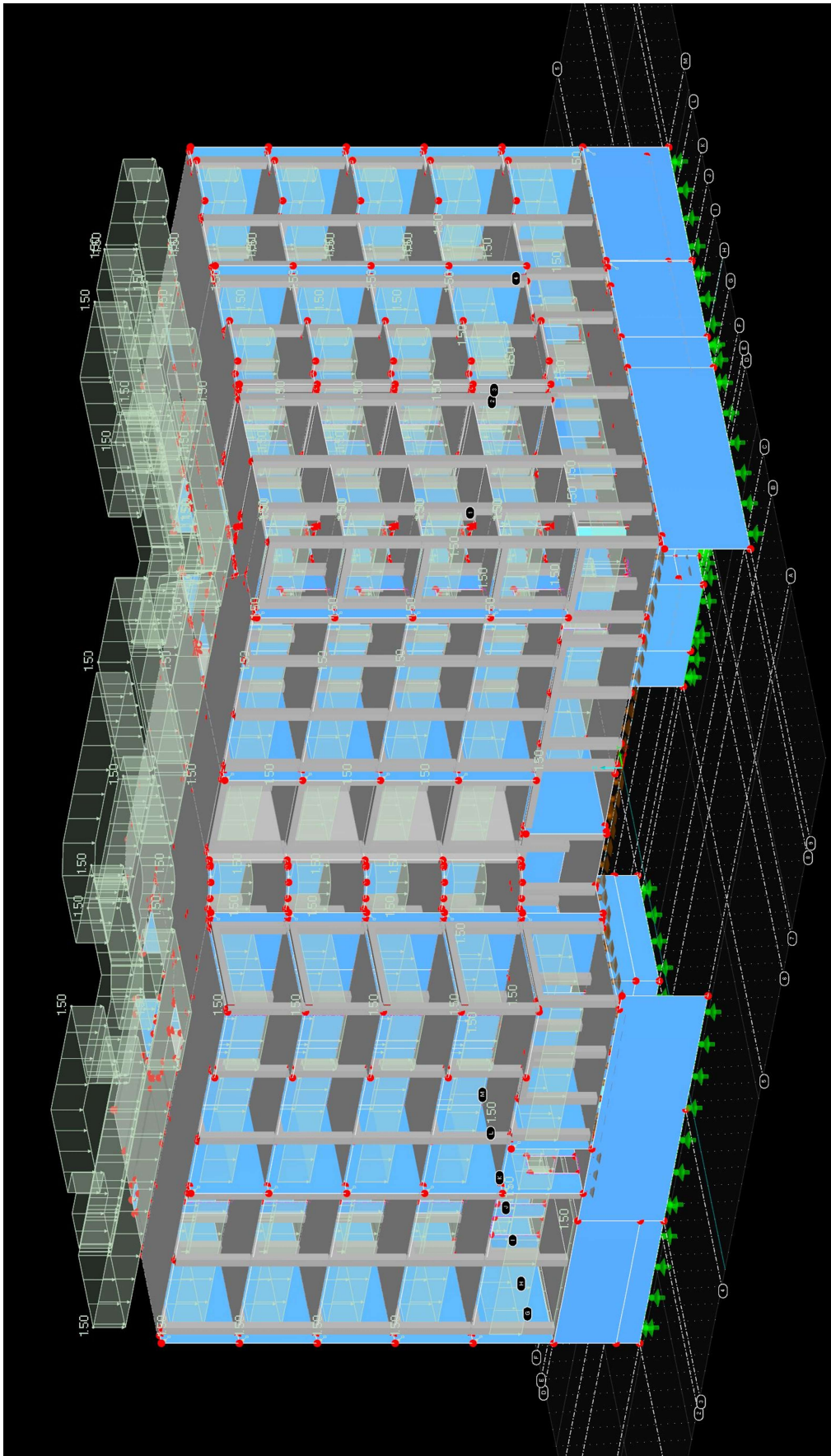
Appendix 2/5
Structural model of the building



Appendix 3/1
Dlubal RFEM generated analytical model
Dead loads



Appendix 3/2
Live load



Appendix 3/4
Wind Load South

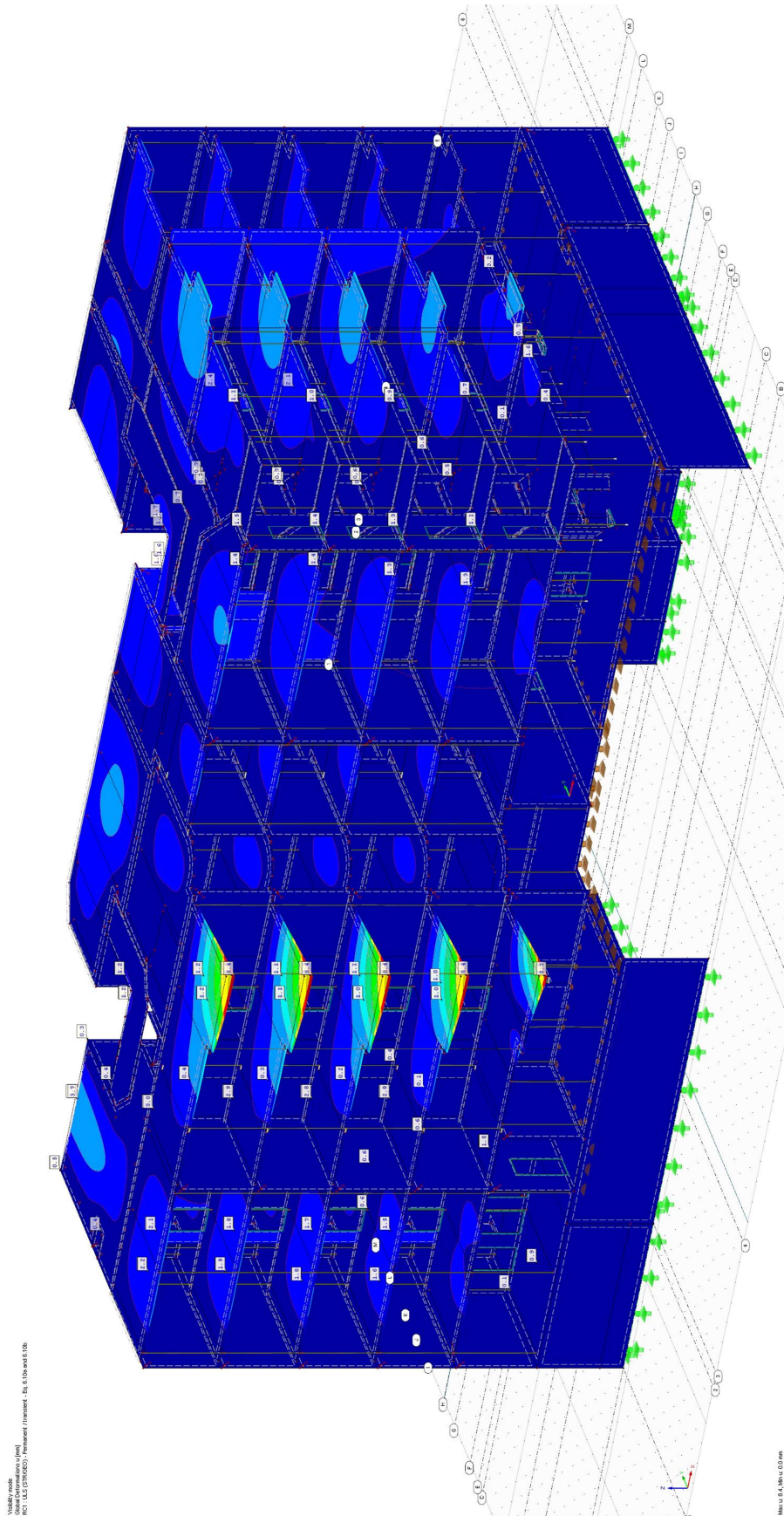


0.27

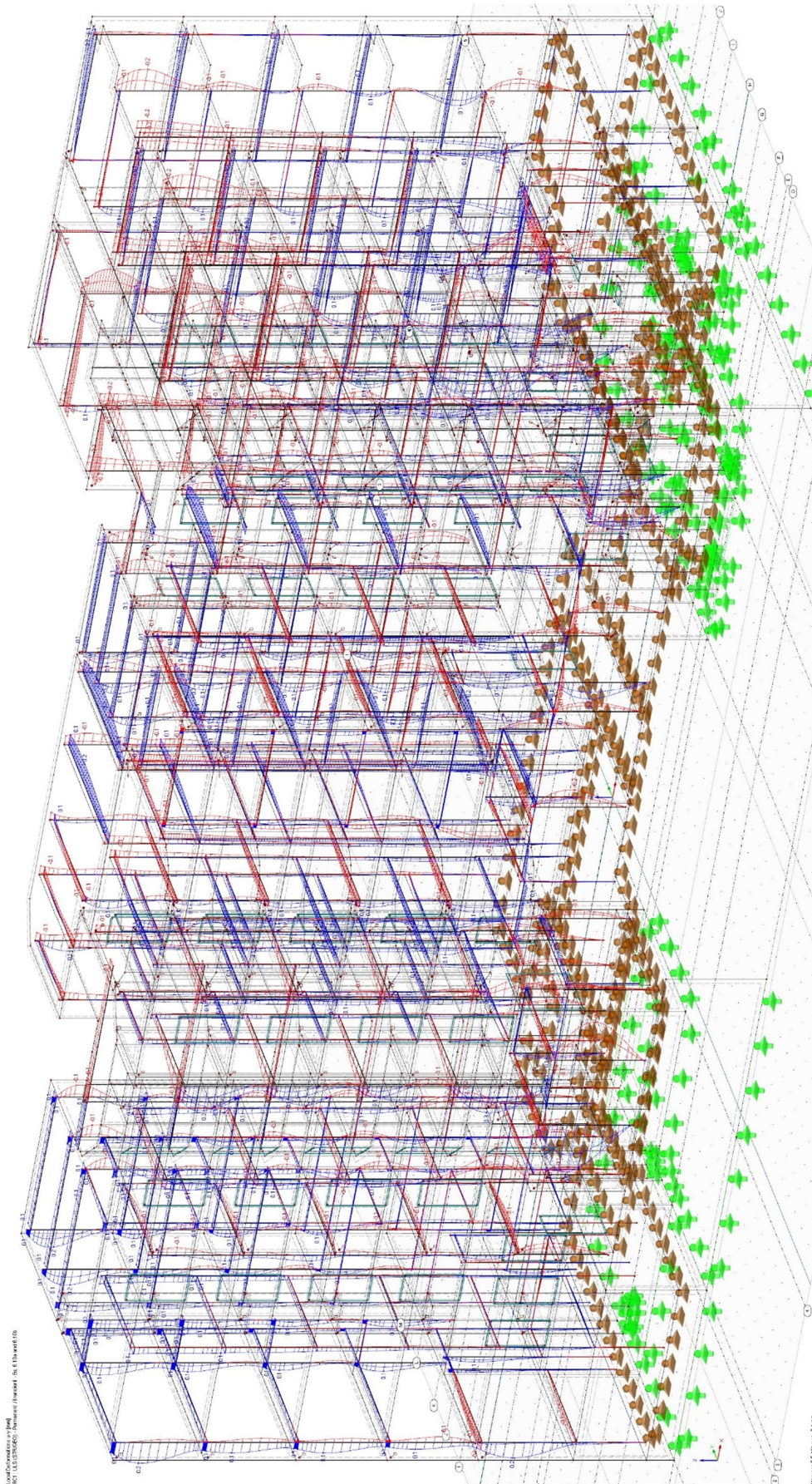
Appendix 3/5
Wind load East



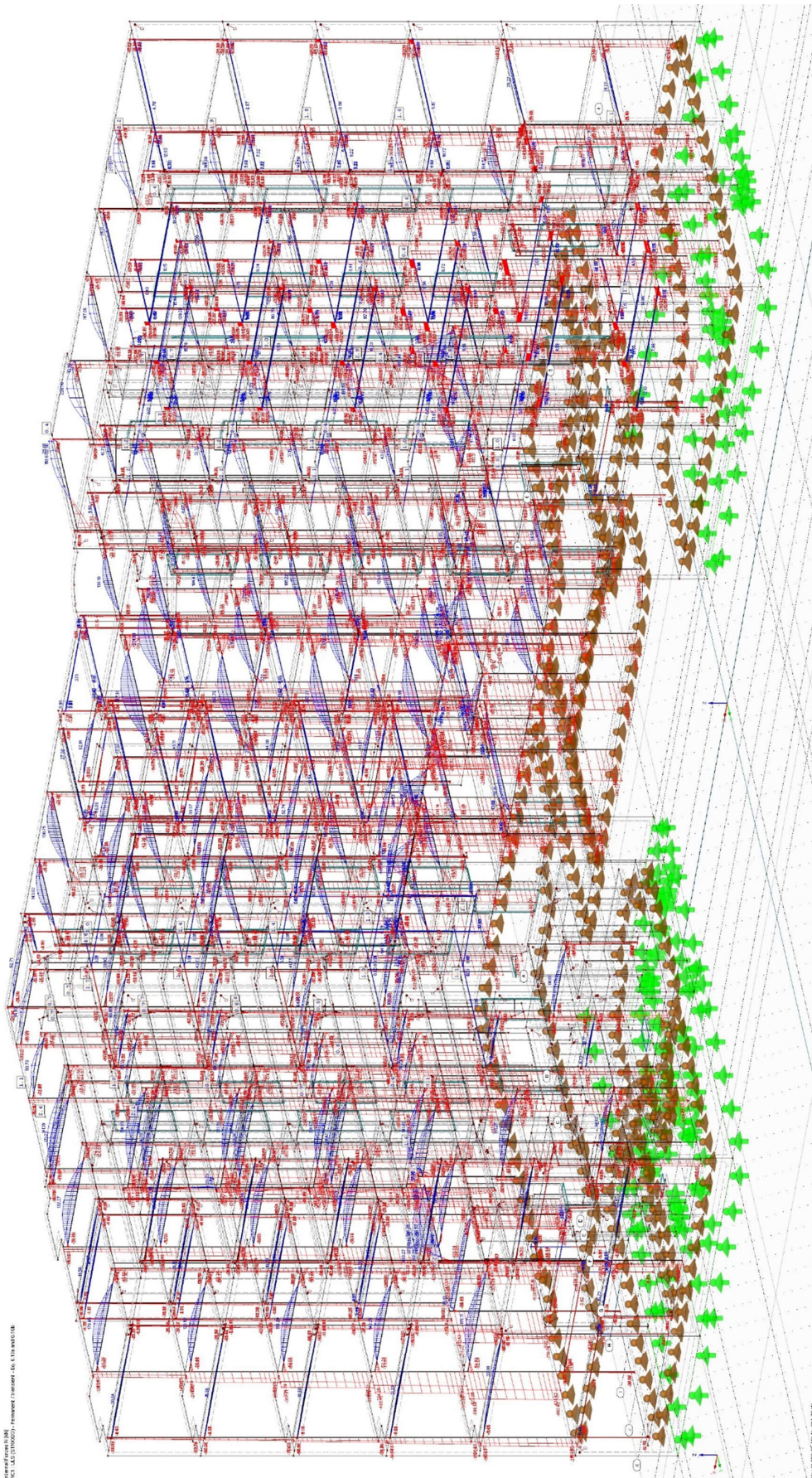
Appendix 4/1
Structural analysis results
Global deformations



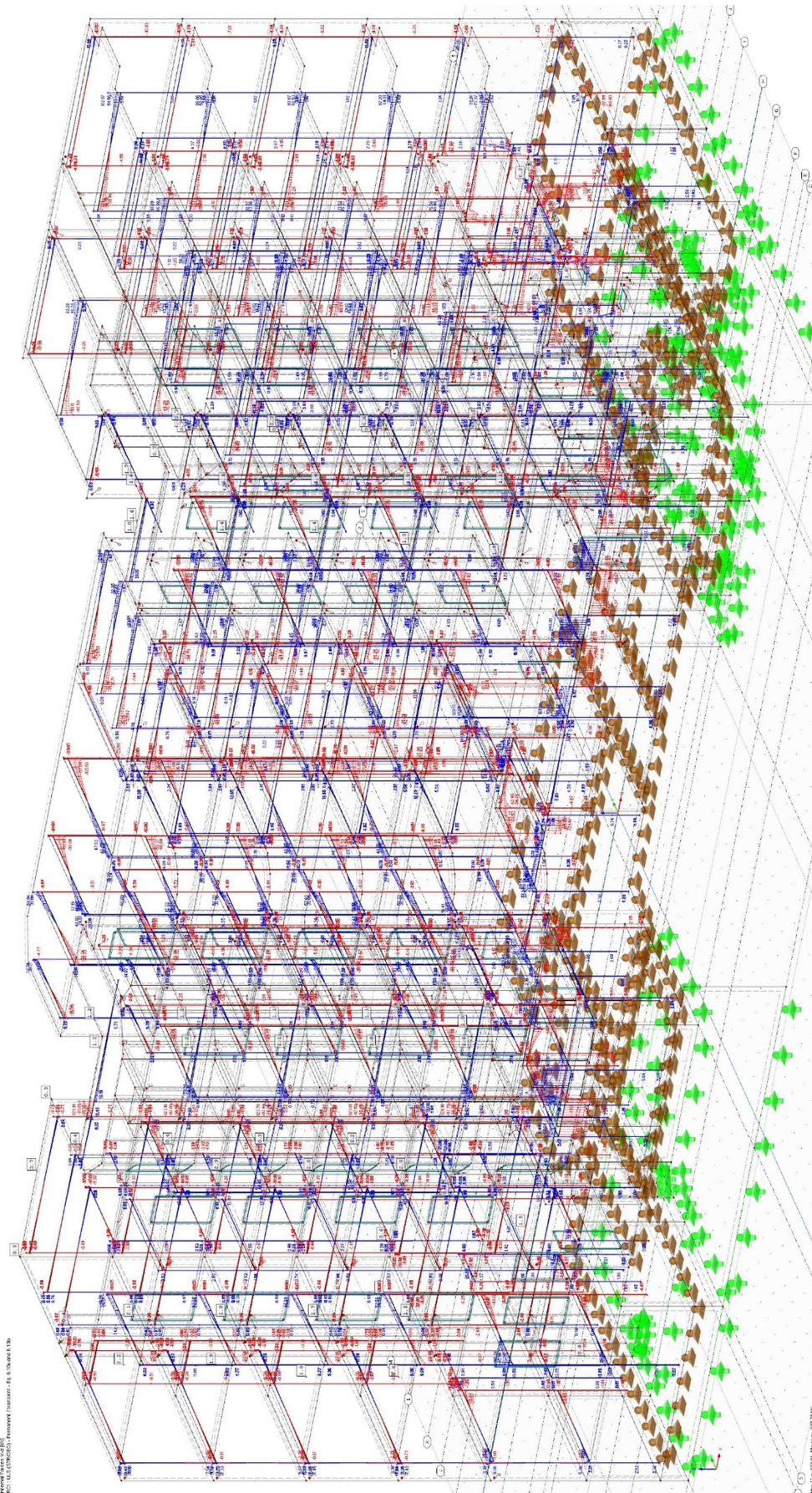
Appendix 4/2
Local deformations



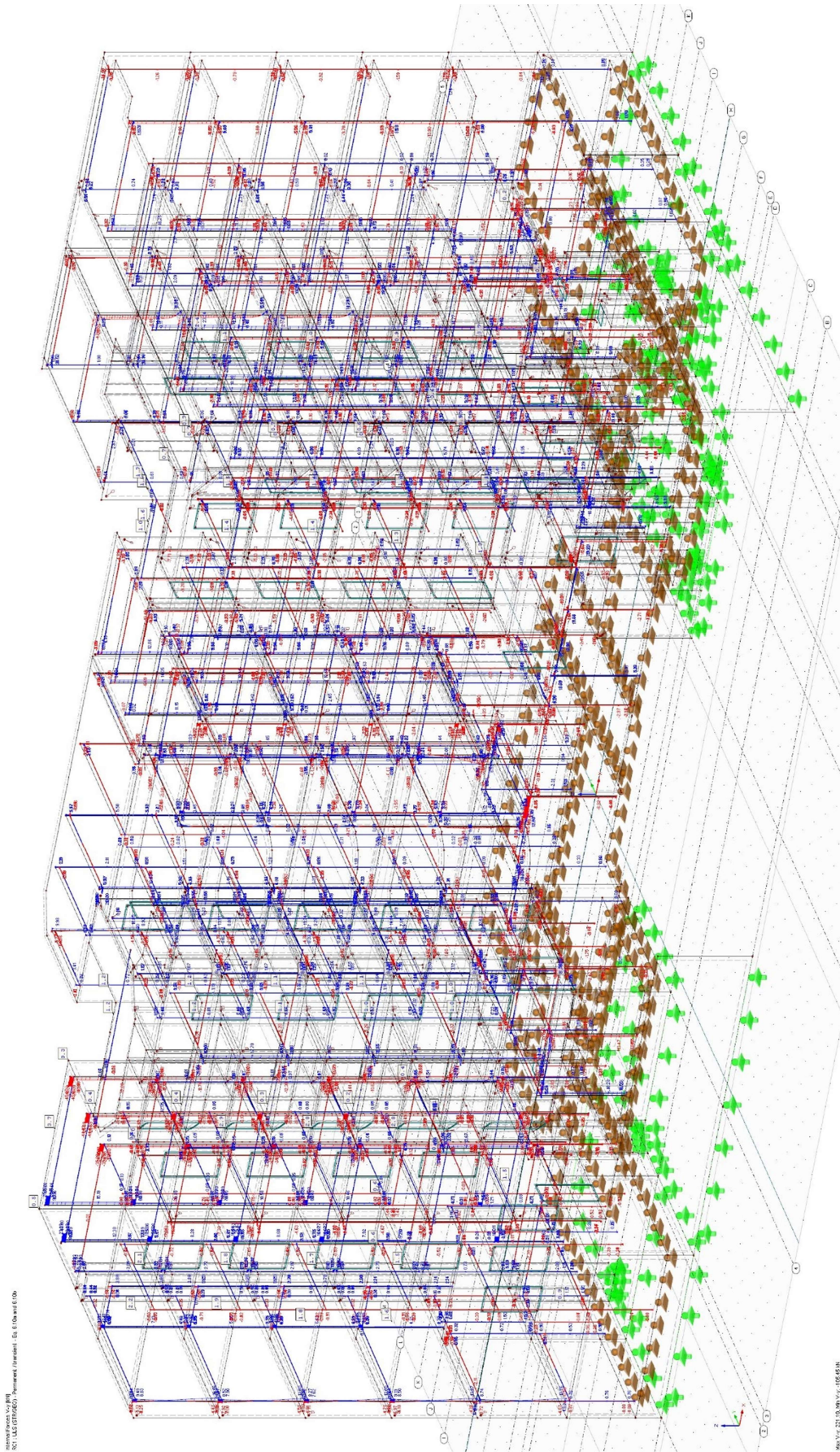
Appendix 4/3
Internal forces



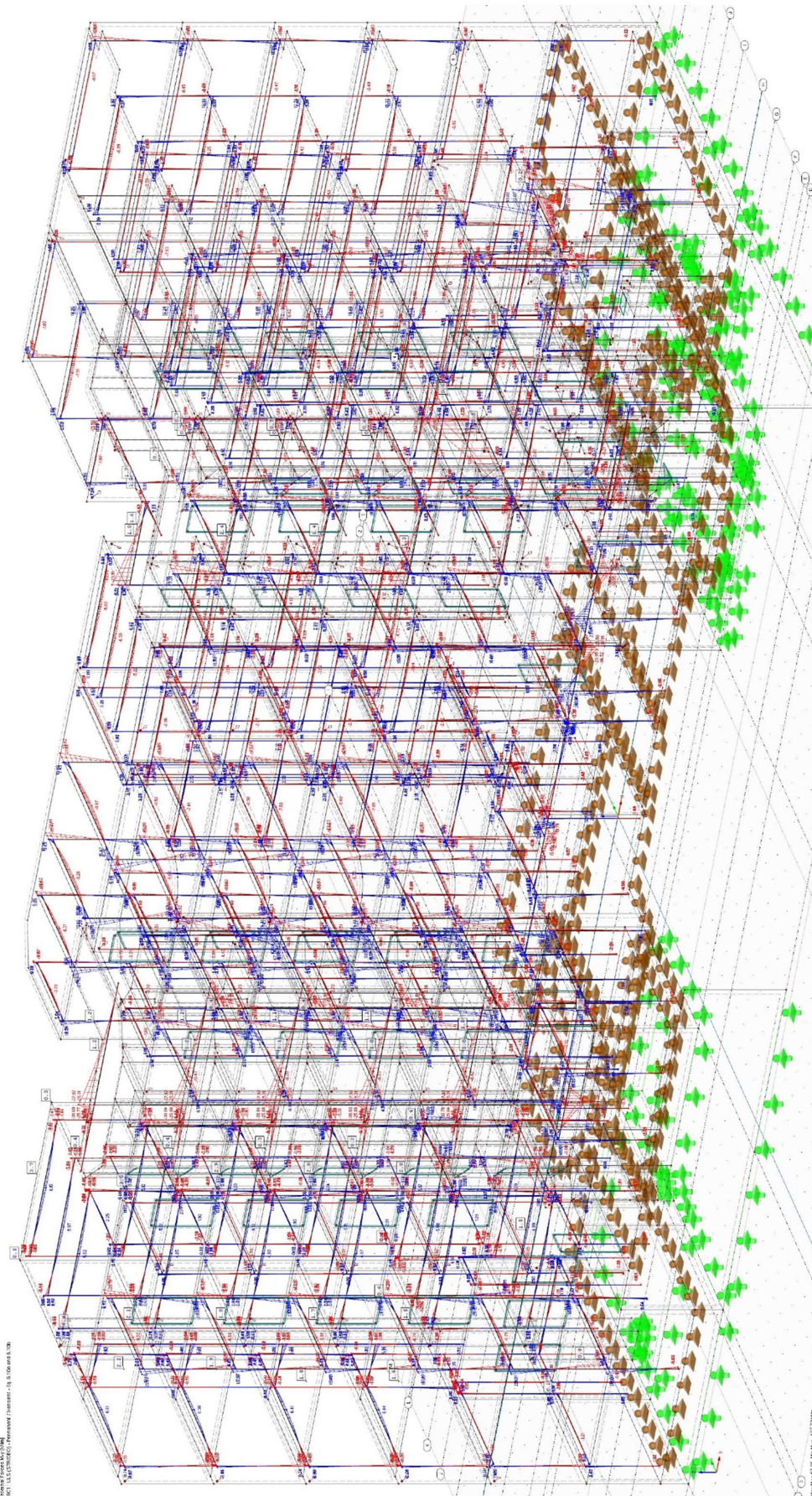
Appendix 4/4
Beams shear force



Appendix 4/5
Columns shear force



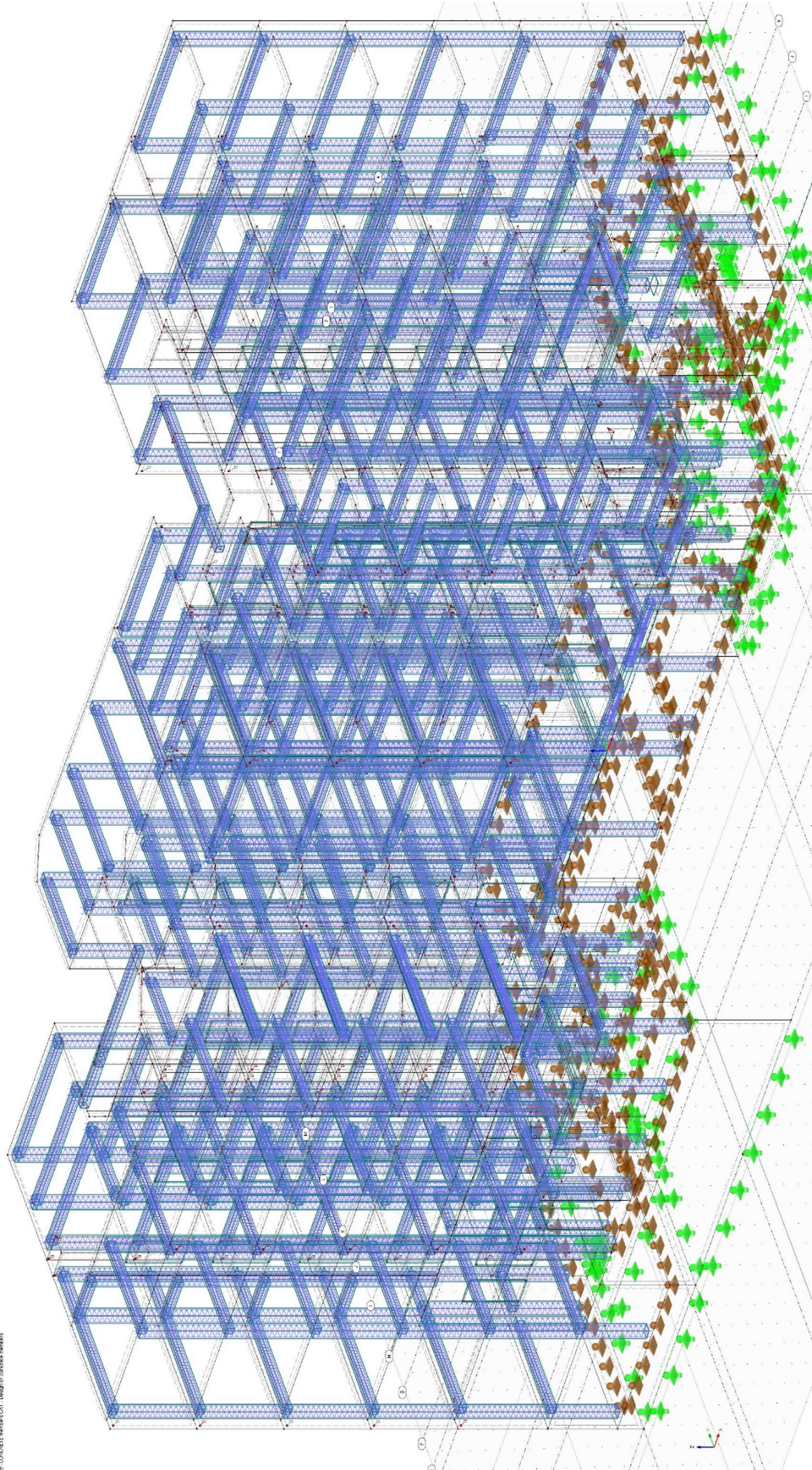
Appendix 4/7
Bending moments



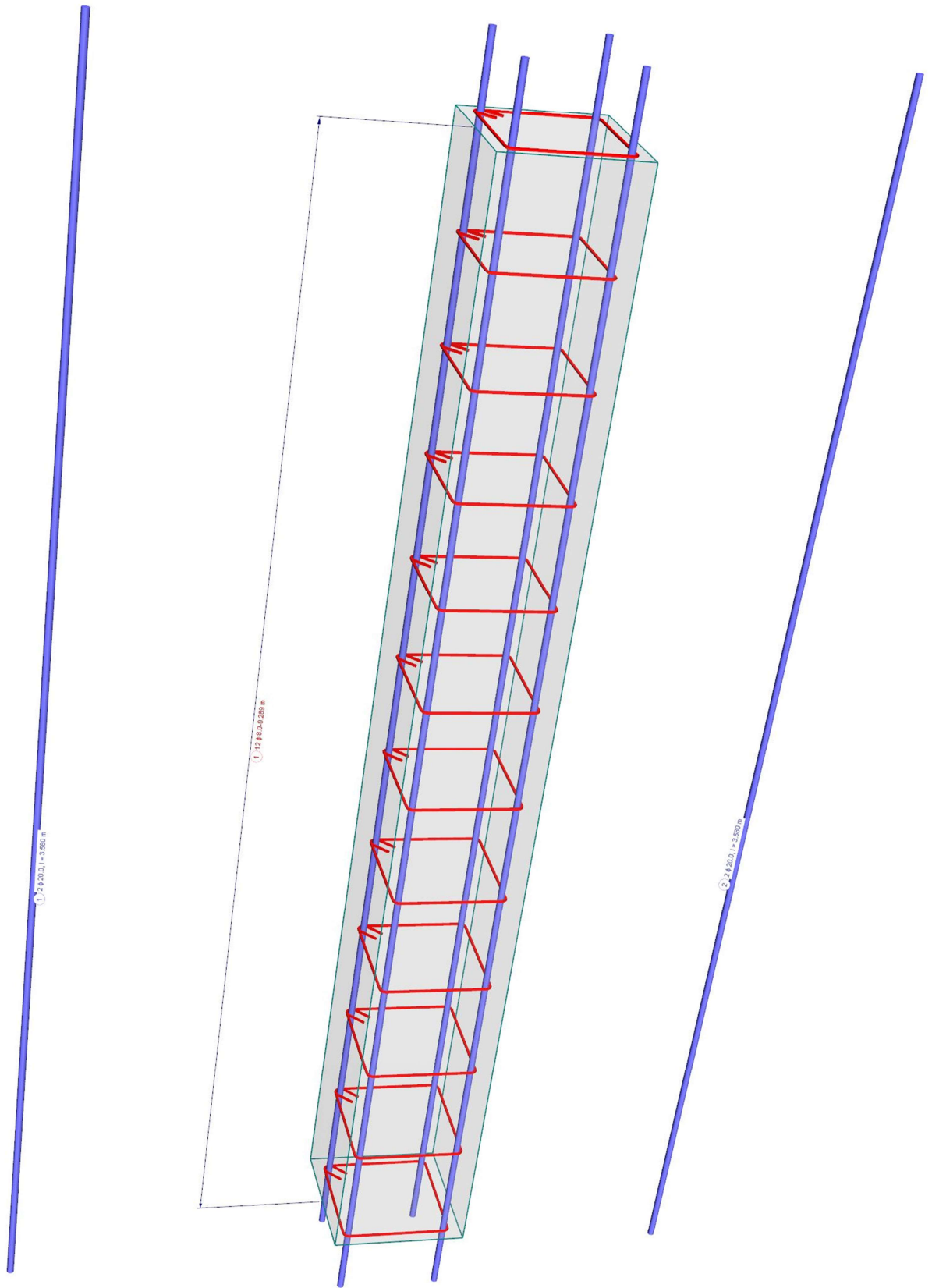
STRUKTUR ENGINEERING (P)VT
SICILIA (ESTRUTTORE) - FORMAZIONE - 19/10/2016 16:53:00

STRUKTUR ENGINEERING (P)VT
SICILIA (ESTRUTTORE) - FORMAZIONE - 19/10/2016 16:53:00

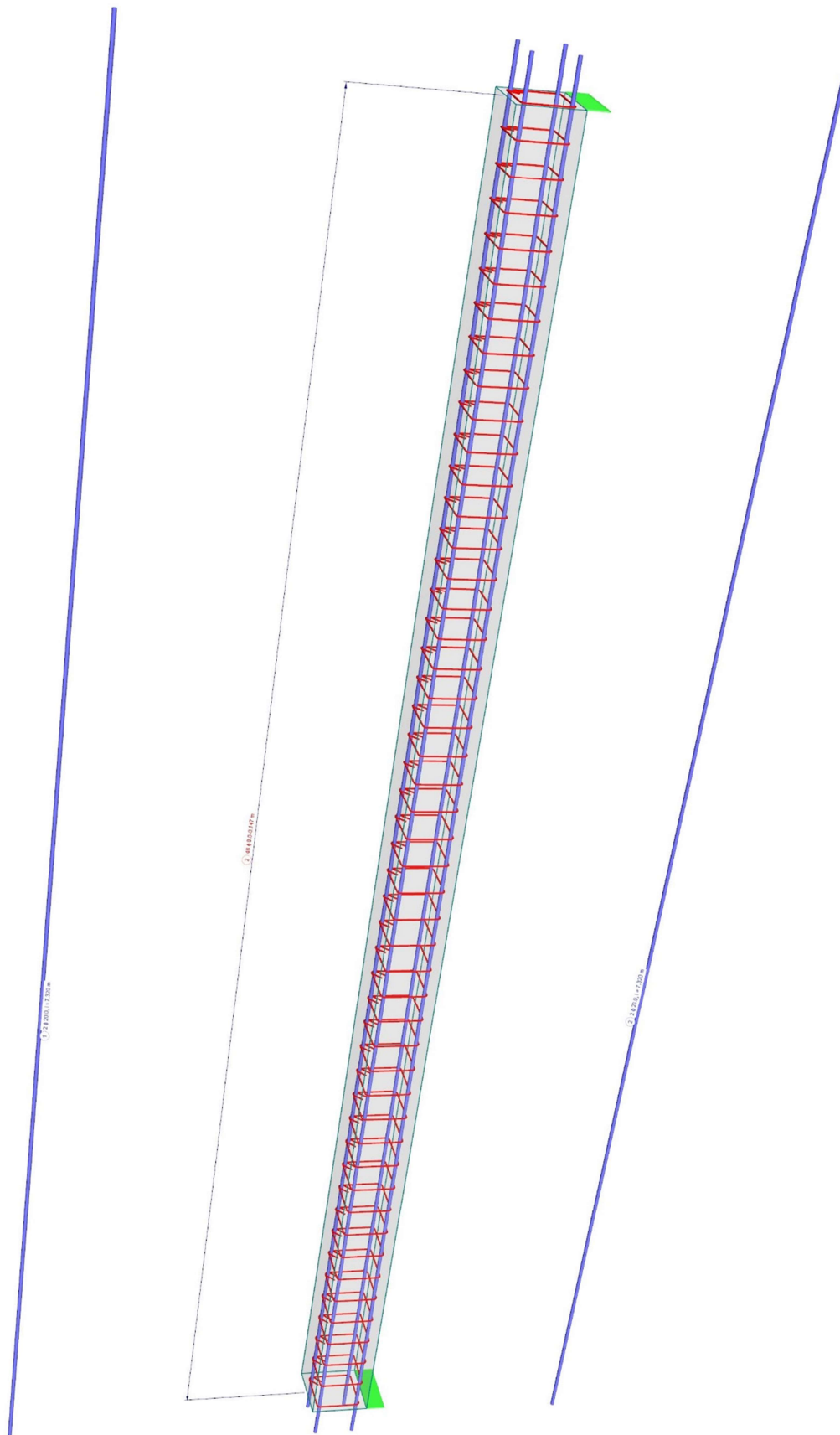
Appendix 5/1
RF-Concrete structural designs
RF-Concrete generated reinforced concrete members

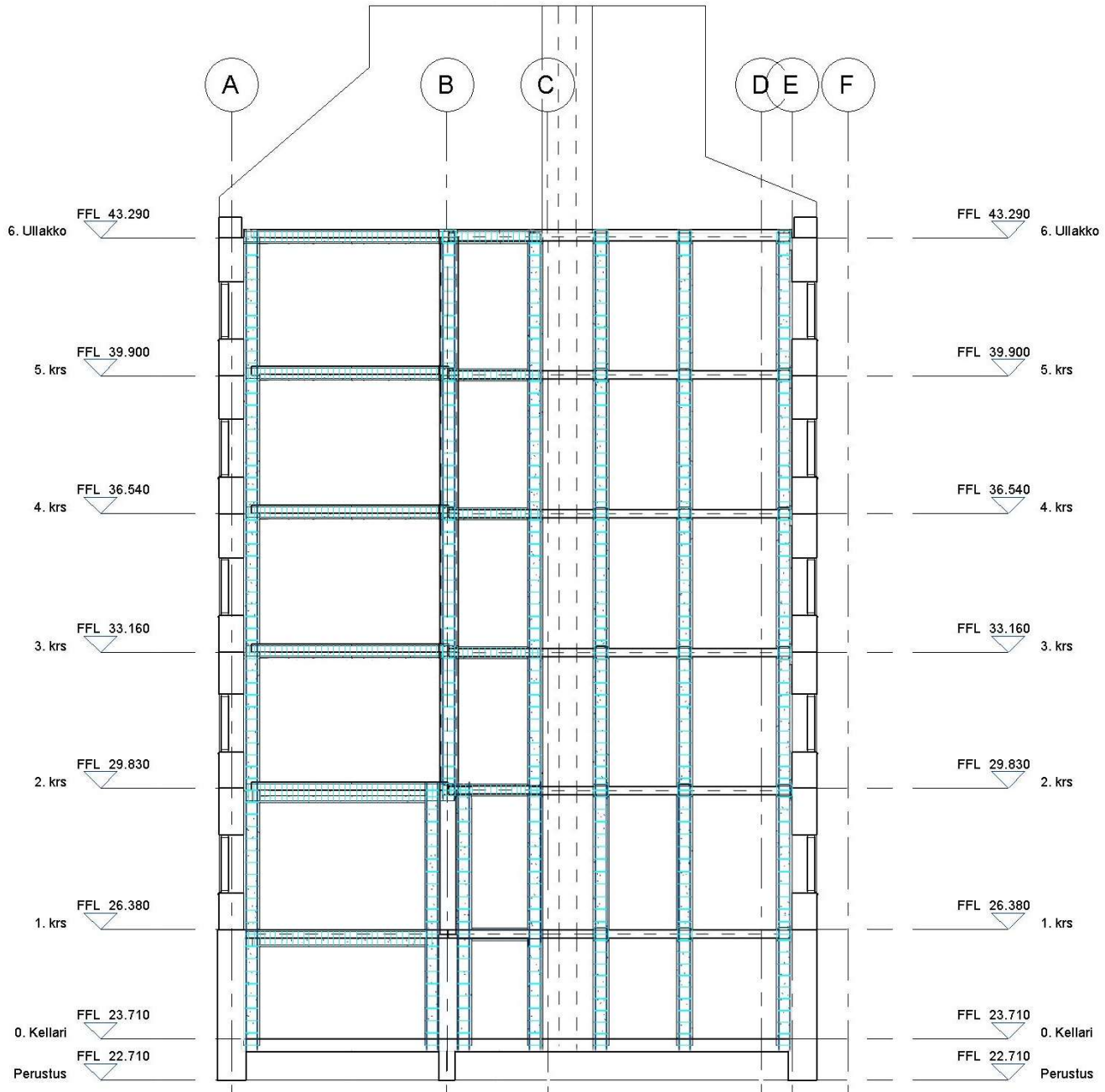


Appendix 5/2
Reinforced column

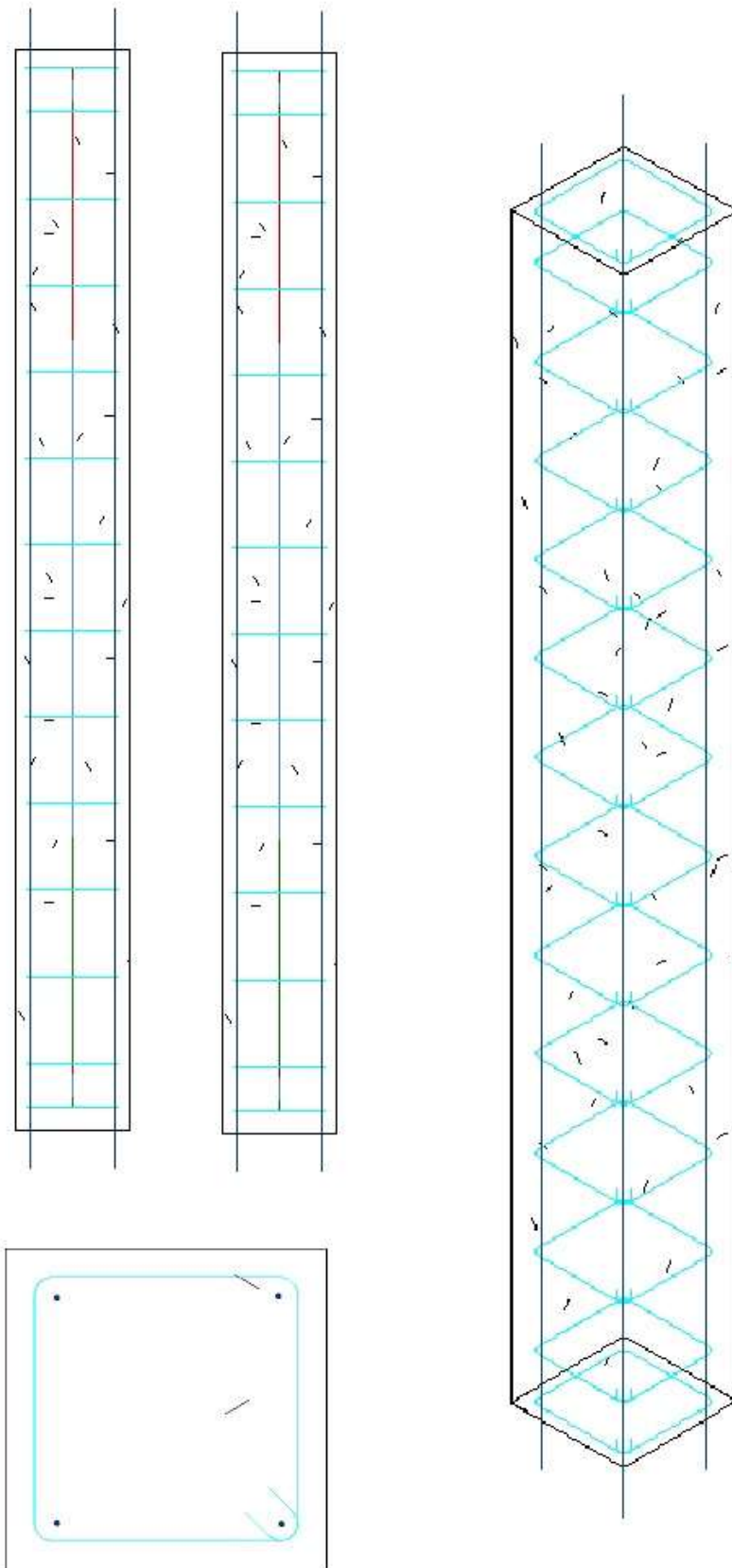


Appendix 5/3
Reinforced beam



Appendix 6/1
Revit reinforced section view

Appendix 6/2
Revit reinforced Column



Appendix 6/3
Revit reinforced beam

