



**FCTUC DEPARTAMENTO DE ENGENHARIA CIVIL**  
FACULDADE DE CIÊNCIAS E TECNOLOGIA  
UNIVERSIDADE DE COIMBRA

# **Construction design of Hotel with Spa using BIM technology**

**Dissertação apresentada para a obtenção do grau de Mestre em Engenharia Civil na Especialidade de Mecânica Estrutural**

**Author**

**Anita Gumiela**

**Supervisor**

**João Pedro Martins**

Esta dissertação é da exclusiva responsabilidade do seu autor, não tendo sofrido correções após a defesa em provas públicas. O Departamento de Engenharia Civil da FCTUC declina qualquer responsabilidade pelo uso da informação apresentada

**Coimbra, June, 2016**

## Abstract

Building information modeling, that means modeling information about buildings and structures. The object model shows the physical and functional characteristics of the building. Model consists of a three-dimensional elements such as beams, columns, walls of which are assigned to them the parameters. Industry foundation classes is a text-format data. this format is used to save files, which can be used in a variety of programs that support BIM.

BIM technology is a technology that gives out a lot of ability, introduces the civil engineering to the another level of design, it allows to create more complex and modern buildings. Its implementation to the company can be challenging, considering the fact of learning new software by the employees, but also to change the style of working. However, the company which do not decide for introduced the BIM in the future will not fulfill the required standards.

In this work would be checked the possibility of using this technology in order to perform the project of building a hotel with Spa, reinforced concrete structure. The work would be divided into four stages. The first stage would be the possibility of sending a architecture model from the ArchiCAD 19 program to the Revit 2015. The next step would be to separate the structural model from the building model and correct the mistakes. Then perform calculations using the Autodesk Robot Structural Analysis Professional 2015 and selecting the appropriate reinforcement. The final step is to update the building model and create the drawing documentation using the available in the program tools.

**Keywords:** BIM technology, the building model, Reinforced concrete structure, Hotel with Spa

## Table of contents

Abstract.....	i
Table of contents.....	ii
List of Figure.....	iv
List of Tables .....	vi
1 INTRODUCTION.....	4
2 BIM - software .....	6
2.1 Introduction .....	6
2.2 IFC .....	8
2.3 Programs.....	11
3 Pre-design .....	13
3.1 The geometry of the building .....	13
3.2 Creating IFC file.....	14
3.3 The building model in Revit and link between Revit files.....	14
3.4 Apply case of loads .....	22
3.4.1 DL1 and DL2.....	23
3.4.2 LL1 .....	28
3.4.3 WIND1 .....	28
3.4.4 SNOW1 .....	31
3.5 Generation of calculation model .....	32
3.6 Combinations.....	34
3.6.1 SGN.....	34
3.6.2 SGU .....	36
3.7 Calculations .....	36
3.7.1 Reinforcements for beams.....	37
3.7.2 Reinforcements for columns.....	40
3.7.3 Reinforcements for slabs .....	42
3.7.4 Reinforcements for walls.....	45
3.7.5 Reinforcements for foundations .....	52
4 Drawings and update the building model in Revit.....	55
4.1 Drawings for beams.....	55
4.2 Drawings for columns .....	57
4.3 Drawings for slabs .....	59
4.4 Drawings for walls .....	61
4.5 Drawings for foundations .....	62
5 Conclusion.....	65
6 Attachment .....	67

---

6.1 Robot note for Beam .....	67
6.2 Robot note for Column.....	71
6.3 Robot note for slab .....	74
6.4 Robot note for Foundation.....	82
7 References .....	vi

---

## List of Figure

Figure 1 BIM life cycle	6
Figure 2 Difference between 3D CAD and BIM	7
Figure 3 IFC connection to different analysis	9
Figure 4 Architectural base in ArchiCAD	9
Figure 5 Creating IFC format	10
Figure 6 Export IFC format	10
Figure 7 Programs	11
Figure 8 Architecture model in ArchiCad	14
Figure 9 Integration with ArchiCAD	14
Figure 10 New project	15
Figure 11 Revit link	15
Figure 12 Options of link	16
Figure 13 Link the Architecture model in Revit	16
Figure 14 Note about any changes	17
Figure 15 Structural model before correcting errors	18
Figure 16 Structural model after changes	19
Figure 17 Analytical model	19
Figure 18 Structural model in Robot	20
Figure 19 The mistakes in building model	20
Figure 20 Releases	21
Figure 21 Cladding	21
Figure 22 Load distribution	22
Figure 23 All loads from Revit	22
Figure 24 Calculated stairs nr 1	24
Figure 25 Calculated stairs nr 2	25
Figure 26 Calculated stairs nr 3	26
Figure 27 Calculated stair nr 4	27
Figure 28 Generate wind loads	29
Figure 29 Value of wind velocity	29
Figure 30 Wind simulation	30
Figure 31 Visualization of wind	30
Figure 32 Information of completed the simulation	31
Figure 33 Value of snow load	31
Figure 34 All cases of loads	32
Figure 35 Options of Meshing	33
Figure 36 The building model with the mesh	33
Figure 37 Reinforcement options	38
Figure 38 Distribution of main stirrups	38

---

Figure 39	Calculation errors	39
Figure 40	Calculations	39
Figure 41	Generated reinforcement	40
Figure 42	Calculated elements	40
Figure 43	Calculated reinforcement	41
Figure 44	Calculated element for lower floor	42
Figure 45	Required reinforcement	43
Figure 46	Required reinforcement for last slab	43
Figure 47	Top reinforcement for roof	44
Figure 48	Bottom reinforcement for roof	44
Figure 49	Shell reinforcement	45
Figure 50	Calculation status for walls	45
Figure 51	Member forces N in direction xx	47
Figure 52	Moments in direction xx	48
Figure 53	Soil loads	51
Figure 54	The moments value of the soil load	51
Figure 55	Scheme of reinforcement	52
Figure 56	Foundation dimensions	53
Figure 57	Calculated reinforcement	53
Figure 58	Continuous footing	54
Figure 59	Reinforcement for continuous footing	54
Figure 60	Reinforcement settings	56
Figure 61	Reinforcement for beam	56
Figure 62	SOFiSTiK app	57
Figure 63	Stirrups for column	58
Figure 64	Reinforcement for column	58
Figure 65	Division slab	59
Figure 66	Reinforcement on the edge	60
Figure 67	Correct bars on the edge	60
Figure 68	Description of bars	61
Figure 69	Reinforcement for walls	62
Figure 70	Dowels for foundation	63
Figure 71	Reinforcement for continuous footing	63
Figure 72	Reinforcement of the building	64

---

**List of Tables**

Table 1 List of the loads on floor 1	23
Table 2 List of the loads on floor 0	23
Table 3 List of the loads for garage	23
Table 4 List of the loads for roof	24
Table 5 The design value of impacts	35
Table 6 The value of the coefficients	35
Table 7 SNG Combinations	35

## 1 INTRODUCTION

The topic of this work is the implementation of the structural project for Hotel with Spa using the BIM technology. Currently, design BIM technology is ever more frequent, in some countries for projects relating to public buildings, this technology is required. It has many advantages for all parties involved in the project. In case of the real design of building the best solution would be to perform the various stages of the project in collaboration between architect, designer and installer. The work should be done on a model with the ability to update it during operation. An important aspect is a good cooperation and appropriate software. Several companies offer different kind of programs using BIM technology, that are able to work together.

The purpose of the work is to familiarize the reader with this technology, its advantages and difficulties they may encounter in the process of including it to the company and what kind of errors can appear during the execution of the project. One of the goals of this work is to check the programs offered by Autodesk and check how they cooperate between them. The company has a number of programs designed to carry out all stages starting from architecture model, visualization and cost estimation. The programs were selected from this company because of the student versions available for free.

The work is based on the building model received from the Cathedral of Architecture, Technical University of Rzeszow and done in part of a thesis by one of the student. This model was made in the ArchiCAD program.

The first step would be to send the architecture model using the IFC format to another program that is Revit 2015. In this program would be separated the structural model from architecture, and if necessary would be done some changes and correcting errors. In this program, it is also possible to perform installations.

The next stage of work would be sending the building model to the Autodesk Robot Structural Analysis Professional 2015. In the program would be checked the analytical model for the design building. They would be applied all necessary load and made manually combinations based on the Eurocodes. Then would be generated mesh and performed calculations. Then for each structural element would be set appropriate parameters for which would be selected reinforcement. If necessary, dimensions of the elements would be changed.



The last step would be update the model in the Revit program, verification of the changes and by using the tools available in the program preparation the drawing documentation.

## 2 BIM - software

### 2.1 Introduction

Nowadays construction projects are more difficult and complex to manage.

Building Information Modelling (BIM) is a technology and process, not only the software, called n-D Modelling or Virtual Prototyping Technology.

BIM not only uses three-dimensional intelligent models but also makes significant changes in the workflow and project delivery processes. BIM represents a new paradigm within the Architecture-Engineering-Construction (AEC), one that encourages integration of the roles of all stakeholders on a project.

BIM supports the idea of Integrated Project Delivery (IPD) which is a new approach to the delivery of the project of integrated people, systems, business structures and practices into a collaborative process to reduce waste and optimize performance in all phases of the project life cycle - programming, design, preconstruction, construction and post-construction as shown in Figure 1.

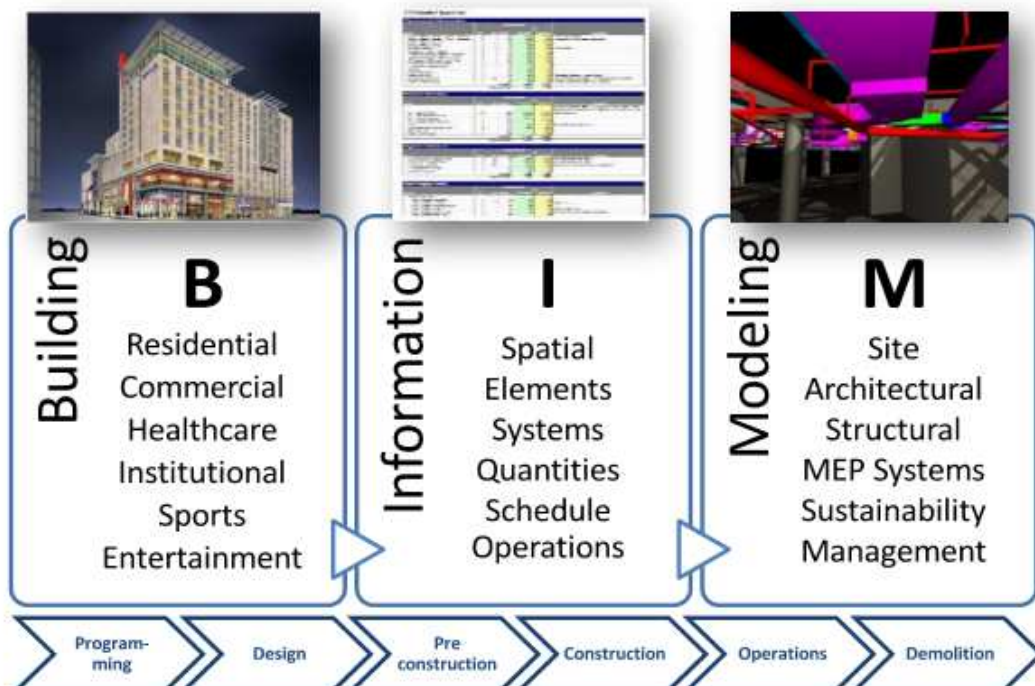
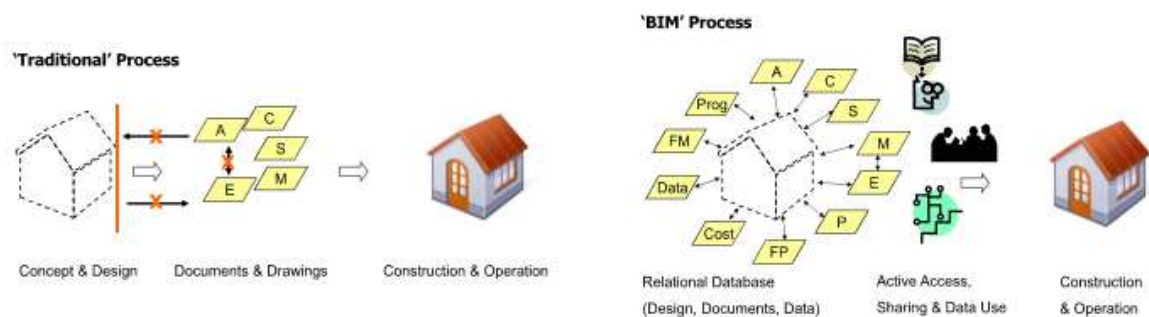


Figure 1 BIM life cycle

5D BIM is understood as BIM that includes, in addition to the 3D model, scheduling information (the 4th D) and information for project evaluation model (the 5th D).

The main difference between BIM technology and conventional 3D CAD is that, the building is describes of independent 3D views, such as sections, plans. Changing one of the previews will need to update and verify the other and this can lead to many mistakes. Azhar (2012)

Furthermore, in case of 3D model elements are only graphic elements such as line, arcs as opposed to BIM, where the objects are defined as different categories of building elements and systems, such as beams, columns and walls. Figure 2 shows the difference in the design process in the traditional way and in BIM technology, which is the lack of cooperation and separating from each stage of the design and documentation process.



**Figure 2 Difference between 3D CAD and BIM**

### **The main benefits for project designers of BIM:**

- better design object associated with a more rigorous analyzes mode by the software simulations and visualizations
- early inclusion in the planned building of its functions in sustainable development in the construction of the building, in order to environmental performance
- improved compatibility and accuracy in the project by means of visual inspection and analysis that leads to better performance documentation
- the possibility of early detection of collisions, crashes, and designing evacuation plans
- faster and more accurate production of shop or fabrication drawings
- creation based on the building model cost estimate and schedule, the possibility of updating at every stage of the work, to avoid costly errors
- early detection of various types of design errors and collisions in the building model
- construction planning and constructability analysis
- offsite prefabrication and modularization
- site safety planning
- better communication with project owner, designer, subcontractors and workers on site

**Barriers to BIM implementation**

- one of the biggest challenges is to raise awareness, education and qualifications of employees
- another problem is the purchase of appropriate software which is very expensive and for small businesses difficult to execute
- the problem may be the lack of knowledge of how best to organize a team to take advantage of BIM
- different software used by the individual participants of the project and the problems related to the transmission of data between them
- the problem may also be a reluctance to co-operation between the project participants, different objectives and incurring their shared responsibility

The advances in smart phone and tablets technology have allowed contractors and subcontractors to frequently use BIM models at the jobsite for information extraction and coordination. Some of the notable BIM apps include BIMX®, Bentley Navigator®, Buzzsaw®, etc. Recently the Autodesk® has started a Cloud-based service, The Autodesk 360®, which allows users to share BIM models in a web environment and perform various tasks in the field such as walk-through, clash detection and preparing digital RFIs.

**2.2 IFC**

Industry Foundation Classes (IFC) is a data format that became the main standard for data exchange. IFC is non-for-profit which is neutral product model for openBIM, used for the building lifecycle and available to all interested stakeholders. It was developed by the buildingSMART in order to facilitate cooperation in the building industry. The main benefits of BIM are possible through sharing information between organizations, IT systems, databases and departments. The users of IFC standard do not need a specific file format or extra product to be able to work on different software. The IFC gives the ability to exchange information about building between CAD tools. The IFC export is include in some BIM software, or available as separate modules.

Figure 3 Shows what stages of the design process could be connected with thanks to IFC format.

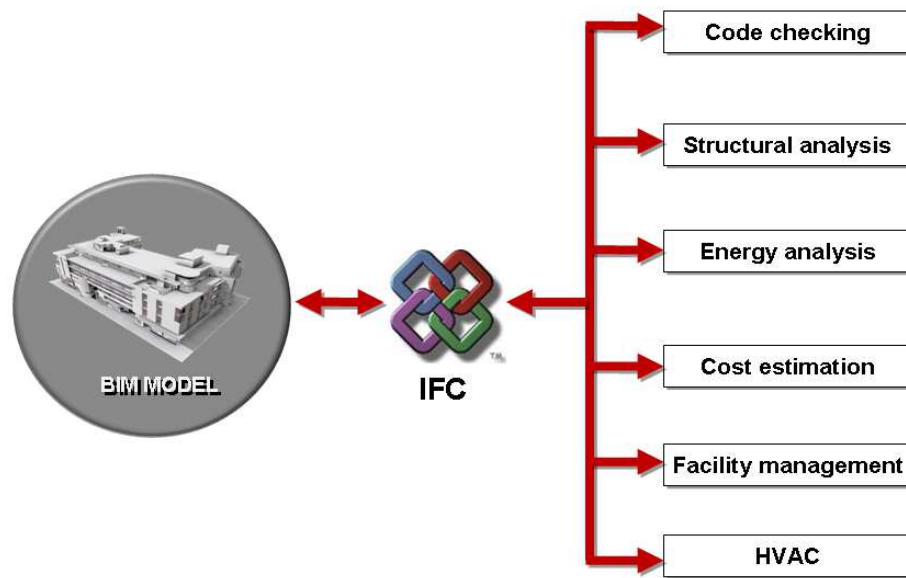


Figure 3 IFC connection to different analysis

The architecture base which is a Hotel with Spa made of concrete from Architecture department show Figure 4.

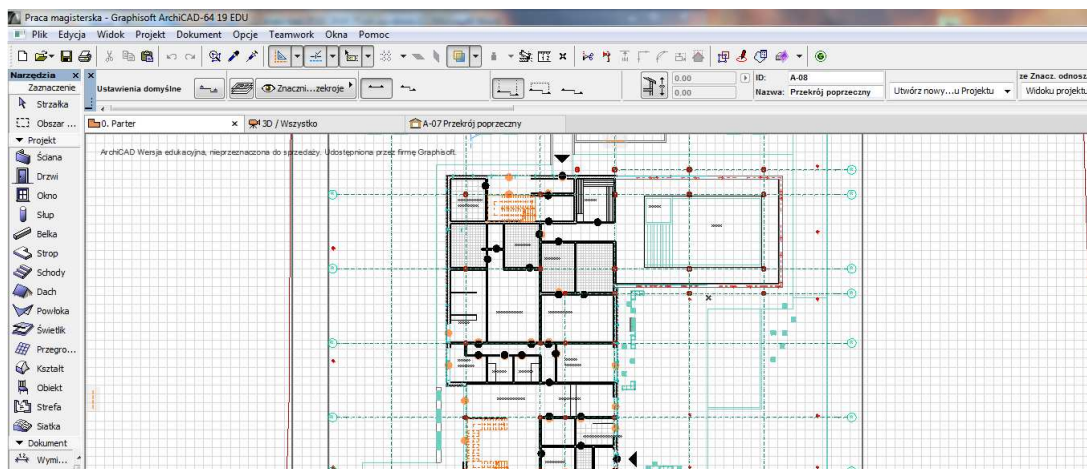
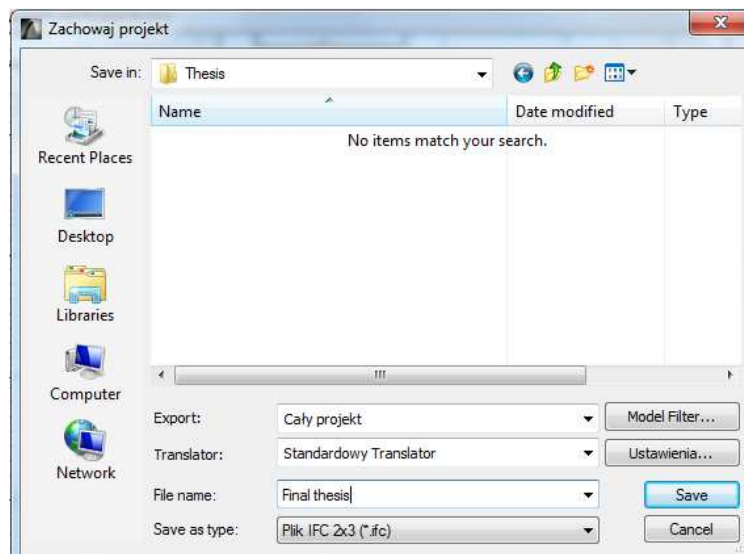


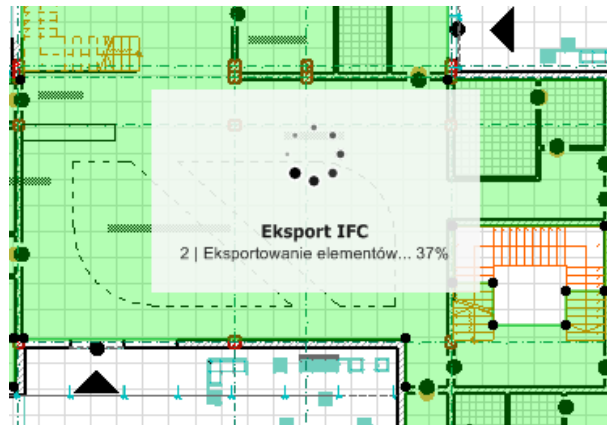
Figure 4 Architectural base in ArchiCAD

To be able to create a structural part of the building. Using IFC format, It is necessary to open model from ArchiCad in Revit. To make the IFC format the project is save us. In window the options are change to export the entire project, standard translator and IFC2x3 type as show Figure 5.



**Figure 5 Creating IFC format**

This makes it possible to open architectural design in another program and continue working on it. This allows to save time and avoid errors that could arise from manual redraw elements. The process of exporting the model to the IFC file shows Figure 6.



**Figure 6 Export IFC format**

## 2.3 Programs



Figure 7 Programs

There is many different programs depending on what area it is necessary to work. Thanks IFC format there is a possibility to work on programs of various companies.

The main task in building model and programs for that: Azhar (2011)

- Architectural model - ArchiCad and Revit allow to design the space model. The architect does not have to draw individual lines, insert immediately ready elements such as walls or windows, which are already assigned to their features. Creating a 3D model also facilitates co-operation with people who do not have knowledge of design, for example investor or prospective customer.
- Structural model - Robot or Tekla permit based on a architectural model to create a structural model. Based on the model are various load applied depending on the use of object and its location. Then the calculated internal forces in elements and matched their cross-sections. Referring to my project, the object is located in Rzeszów and is intended for the hotel with reinforced concrete.
- MEP model - Revit allows to design a ventilation system in the building and how best to lead them. This ensures to avoid collision with structural elements.
- Visualization - to create 3D renderings can be used ArchiCAD or Revit, which allows us to see how the object will look like in reality, if it fits to the environment. The investor can see the finished building before work begins on the site and, if necessary, make changes in the project in order to avoid later costly fixes.
- Fabrication/shop drawings - it is easy to generate shop drawings for various buildings systems using for example Tekla BIM or Allplan. There is no need to create additional separate drawings, data object are sent to the devices performing prefabricated

elements which provides more accurate their performance. In case of any corrections automatically updates the model.

- Code reviews -a process where software check the building model against any mistakes. It can be used by fire departments and other officials to review of building projects.
- Cost estimating - BIM allows for quick and efficient creation of the project cost estimate and bill of materials that are needed. In case of any updates, changes are applied automatically saving time on manual corrections. ZuziaBIM or Navisworks software may be used for this type of activity.
- Construction sequencing - Scheduling is an important part of every project. It allows to plan of each delivery of materials to the construction site and individual work. This allows for earlier submission of orders and execution of prefabricated elements, so that materials may in time arrive on site and also avoid delays. To create timetable we can use the same programs as the cost estimation. This allows for distribution of the total investment cost during the period.
- Conflict, interference and collision detection - building models in Tekla are created in scale in 3D space, all the individual components can be tested due to their interference. This ensures a better management of the space and ability to choose a more economical solution.
- Forensic analysis - based on a building information model can be in the design stage analyze the possible failures and leaks. Designed in accordance to current applicable standards fire zone and emergency exits.
- Facilities management - Programs such as ArchiCad allows for space planning. We can also perform renovations and maintenance operation.



## **3 Pre-design**

### **3.1 The geometry of the building**

#### **Overall description**

The subject of my master thesis is the architectural model made available by Patrycja Matysik. The object of my work is a hotel with SPA located in Rzeszów, Poland. The hotel has a diverse arrangement of functional and spatial. The facility includes guest rooms and common part, the environment is also intended for use, there is a small swimming pool and a place to rest. The architectural form of the building is modern and adapted to the environment. The hotel has various heights. The design of the building consists of two parts, one is a two-storey, the second is a five storey, L-shaped and two open courtyards.

In the project included parking places also for disabled people and underground car park which is located directly under the building. The facility includes shafts communication. The entire building is designed with constructing of reinforced concrete.

#### **Walls**

In the garage are reinforced walls 36cm thick. In the rest of building outer walls are made of ceramic blocks having a thickness of 25cm and a mineral wool thickness of 10cm. Part of the facade is made of frameless glass facades attached to the aluminum profile and architectural concrete panels with a thickness of 5cm. In the building there are three types of internal walls. Made of ceramic blocks 20cm thick, drywall filled with mineral wool 20cm thick and glass walls.

#### **Structures**

The remainder of the building is in construction of post and slab, where the column spacing is 8,30m. The columns are of size 40x40cm, have also been added by me in the area of the holes on the vertical communication beams of size 50x40cm and a ceiling thickness of 30cm. The whole structures is made of reinforced concrete.

#### **Swimming pool**

The building has a swimming pool whose walls are made of waterproof concrete with an appropriate arrangement of layers, according to the used technology with a thickness of 30cm.

### 3.2 Creating IFC file

The first step is to prepare the Architecture model received from architect. The figure 8 shows the model before making any changes opened in ArchiCad 19. For better visibility and faster work some elements which are not necessary in structural model are hidden, such elements like a furnishing inside the building, garden furniture, lamps and landscape. Then the building model in ArchiCad is save us as IFC file, which allows to transfer the model to another program.



Figure 8 Architecture model in ArchiCad

### 3.3 The building model in Revit and link between Revit files

The next step is to open an IFC file in another program. In this thesis it would be Revit 2015. This program allows for the preparation of the analytical model, improving all kinds of errors and send the model to another program to perform calculations.

The program includes tools that allow collaboration between the designer and architect, which is an important aspect of working in BIM technology. It enables easier collaboration and the ability to work on one model. Also the way in which the model is presented is much more transparent and easier to understand, even for those who have no knowledge of design.

The IFC file can be open in Revit program by clicking Improved IFC Import - ArchiCad Connection on the Add-Ins ribbon as show Figure 9. Then in the new window, select a previously saved IFC file and open it.



Figure 9 Integration with ArchiCAD

The whole building model from ArchiCad is in this program, because of not having the library of element from the Architect some elements do not have the appropriate properties. The program sees some objects as elements but not read their properties. Clicking on the column receive information about its dimensions and the material from which it is made, but it is not possible to change the properties. Also the original file is too big and complicated. Working on Architecture model in weak efficiency of computer is too difficult.

To accelerate the work was created the link between Revit files, Architectural and Structural Model. This allows to monitor the elements in both files. When it is created a new Structural Model, it is possible to copy only elements which carry load, like floors, walls and columns. The copy elements can be the same in both files or we can in options change them. Designing new elements will be monitor with elements in Architecture model. The monitoring elements help in better cooperation between architect and constructor. If one of them change something in the model, we will get information about that. It is possible to ignore that message, follow the changes or create the comment.

To create the link it is necessary to open a new Structural Template in Revit program as show Figure 10.

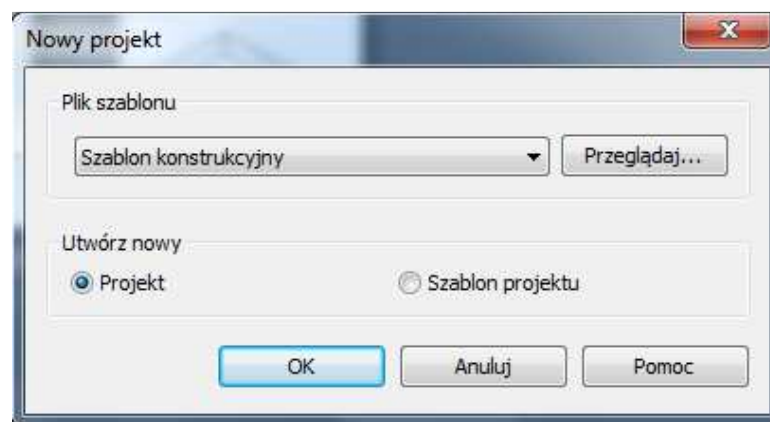


Figure 10 New project

Then open insert ribbon to create link with Revit.

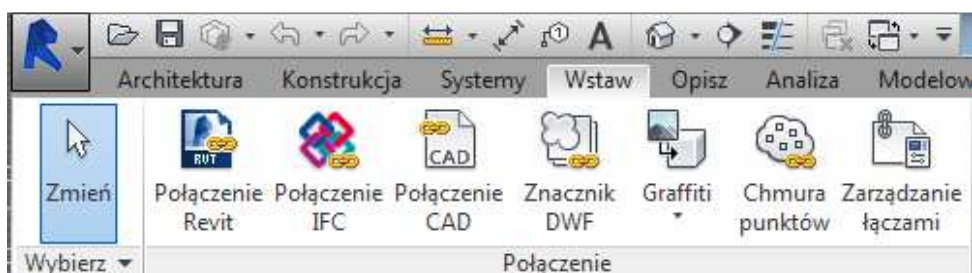
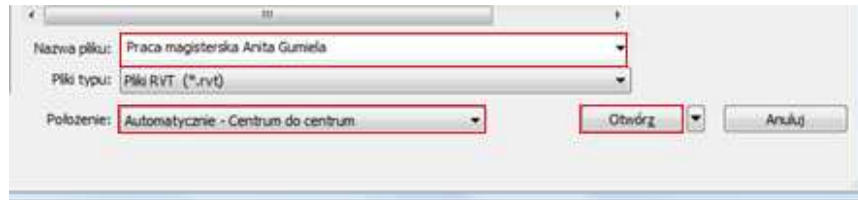


Figure 11 Revit link

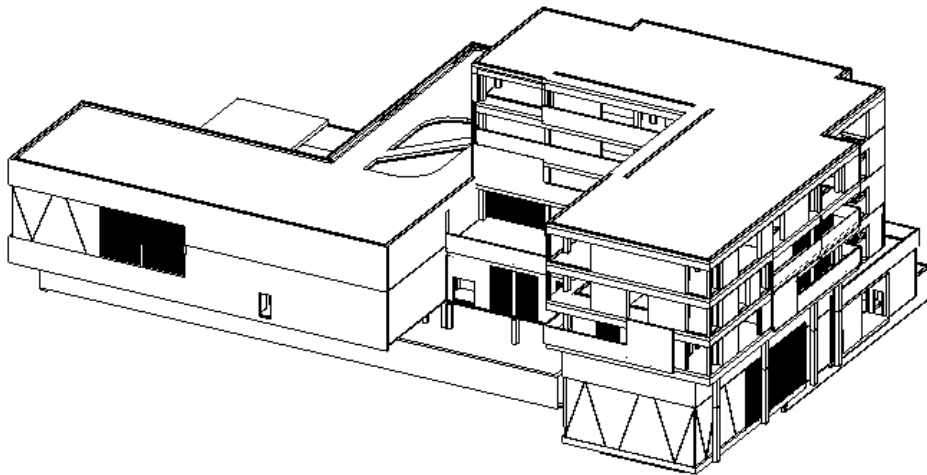
Select the Architecture model in Revit and change the Positioning to Auto - Center to Center as shown below. This means that a included file, coincides with the center of the new project.



**Figure 12**Options of link

Creating links helps with faster working. The size of the file is much more smaller than the original one.

As shown in Figure 13 Architectural model was transferred in full, all the most important elements that are needed for further work can be seen in the link. Therefore it would be possible for them to copy. Elements such as the area, the furniture is hidden because they are not necessary.



**Figure 13** Link the Architecture model in Revit

Directly after create link model from Revit, not all elements are visible, because some elements are hidden in settings. To be able to see them, select elevation East.

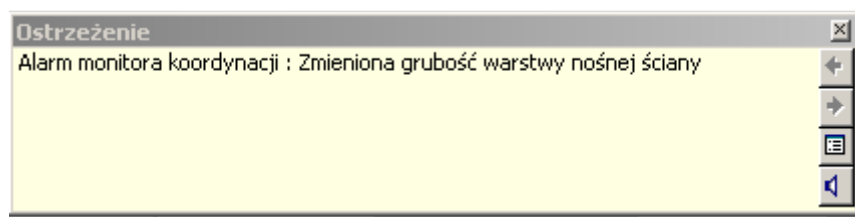
Next step is to select collaborate ribbon, then Copy/Monitor and select the architecture link, that would be a blue frame around the building model.

In options it is possible to change the properties of the elements, add some prefix or suffix to Name of Levels, Grids. For the columns it is selected to copy the original type. For the floors there is selected a new type of the floor, made by concrete of 30cm thick. It is change because in original model omitted the different layers of the floor like mineral wool which is not necessary in structural model.

The first step is to copy elements like levels, grids to create all floor plans for the model, after creating levels View ribbon, Plan Views and Floor Plan and select all new levels. Some floor plans are already in the Revit file. For this levels it is necessary to change the height and then create monitor between new floor plans and with Architecture model.

Now using that model it is possible to copy elements which are necessary to create the structural model. In options It is selected for all columns the size 40x40cm and made from reinforced concrete. The elements will be monitor in both filers. If something become changed in one model, we will get the information in another one, what was changed. It could be accept that changes or ignore. The designer can create a note for another users about the changes. That helps for better cooperation and avoid mistakes in the future. For all slabs It is chosen thickness of 30cm, the original one have 45cm including all layer, It is select only concrete. After creating all elements in options, clicked that the elements are structural to be able to send the model to Autodesk Robot Structural Analysis Professional.

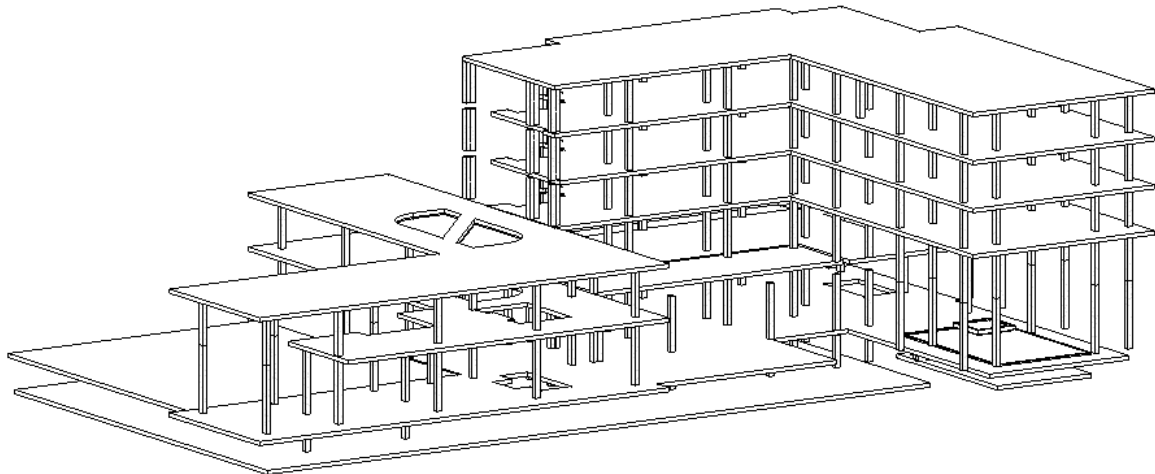
Because of change the thickness which is different, some of elements are not connected. Using Revit tools the columns are attach to the floors. With any changes with Monitor elements, there is the note about it. Shown in Figure 14 the Warning informs us about changing the length of the columns or about changing the thickness of the structural elements.



**Figure 14** Note about any changes

To facilitate the work on the new model, if necessary, it is possible to turn off the visibility of the link, by selecting the View ribbon. Next Visibility/Graphics, to change the view select the Revit Links ribbon and turn off the visibility.

As shown in Figure 15 The copy model from Architect has some errors like lack of some columns, not the exact line of slabs. Some elements are not connected. Next step is to correct all that mistakes.



**Figure 15 Structural model before correcting errors**

Columns that were outside the building have been extended to the same depth as the columns in the garage due to the zone of frozen soil.

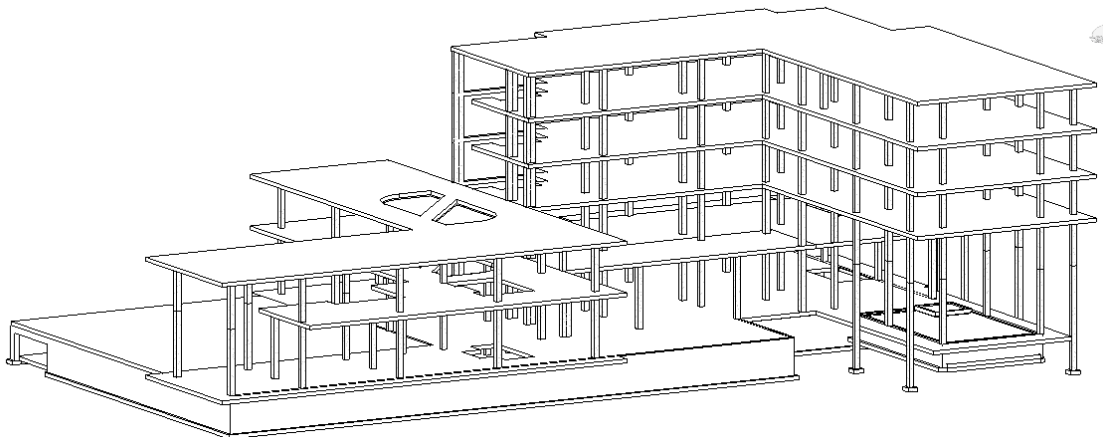
For each of the columns has been added foundation of dimension 30x80x80cm. Next all of them are monitor with foundation from architecture model.

Under the walls which are located in garage were added combined footing of dimension 90x30cm and monitor the same way like the foundation for columns.

They have been added an additional beams around the holes for the stairs in order to strengthen the structure. These elements also provide support later under the walls and support under the stairs.

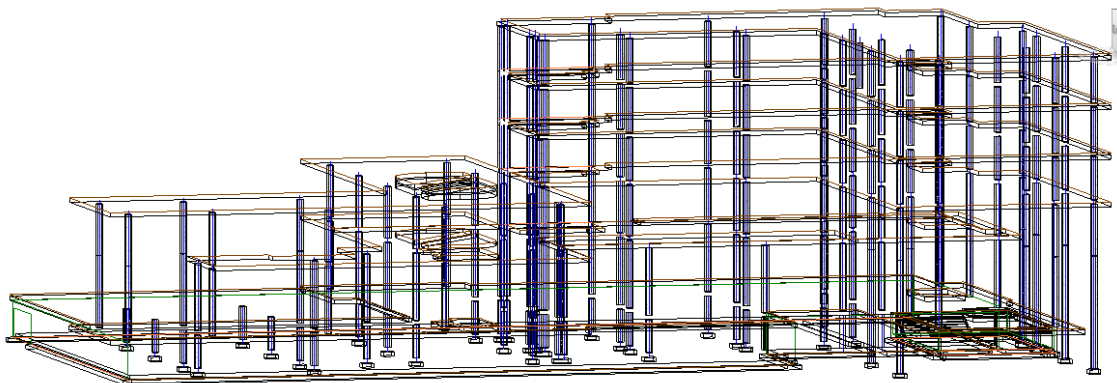
In the original model some walls are made from bricks, glass or drywalls. In new model that elements have been omitted, because they are primarily designed to separate the rooms. They do not represent structural elements and do not carry loads.

The Figure 16 shows the building model made in the Revit program after all the changes. As mentioned above was changed line for some of floors, added additional structural elements and changes the dimensions of some elements.



**Figure 16 Structural model after changes**

After the correction of the structural model it is possible to verify the analytical model of the building. To incorporate the analytical model, click the icon located on the bottom bar settings. As shown the Figure 14 the model is not correct, some of the elements like columns and walls must be extended. This can be done by connecting corresponding points at the tips of the an analytic model by selecting Analyze ribbon, next Analytical Adjust. After clicking that we receive the model on which it is able to make changes.

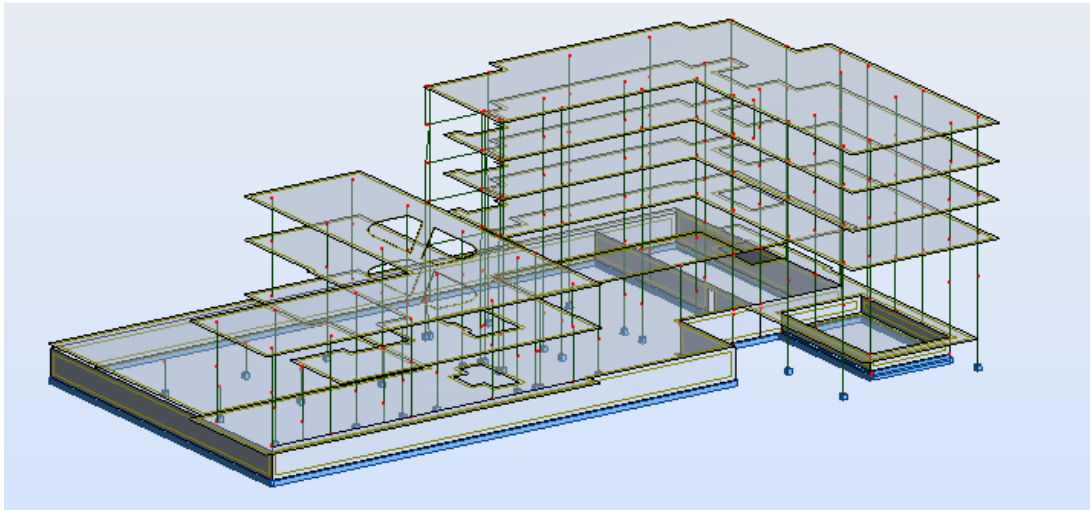


**Figure 17 Analytical model**

After correcting any errors in the model and check it, the next step is to create a integration between Revit 2015 and Autodesk Robot Structural Analysis Professional 2015. To create connection the building model is first sent to the intermediate file (.smxx). In Autodesk Robot Structural Analysis Professional 2015 it is open a new project - Building Design. To create integration in Robot using that file in this version of the program it is necessary to select Add-Ins ribbon, next Integration and Autodesk Revit Structure. In new window select Send to the intermediate file (.smxx) and select previous file. While direct sending model from Revit

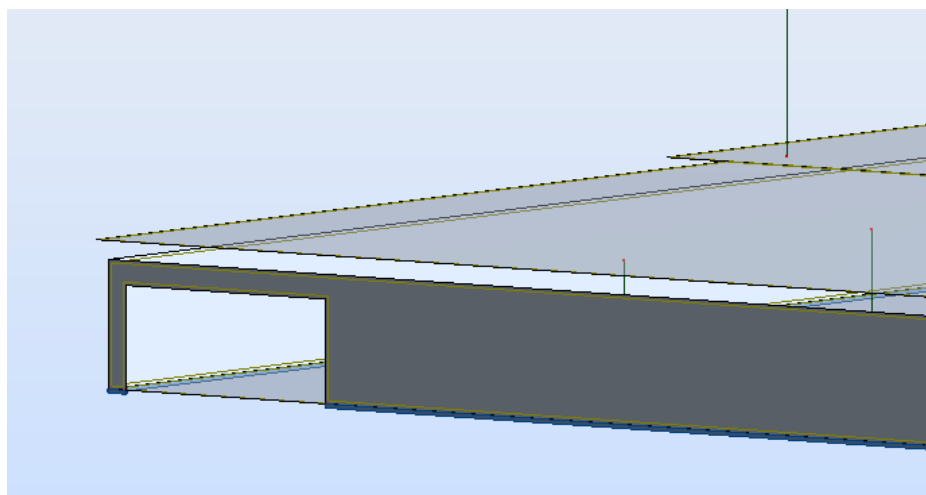
program to Robot program appears error. After the creation of the connection between the two programs it is possible to transfer the building model to perform calculations.

Figure 18 shows the building model after sending. All elements which were in Revit program selected as a structural elements are in Autodesk Robot Structural Analysis Professional. The ability to sent a model from one program to another allows for faster project execution. This also allows to avoid unnecessary errors and better cooperation.



**Figure 18 Structural model in Robot**

Before executing the next steps it is important to verify that all components are properly connected. As shown at the Figure 19 some of the elements are not connected, the foundation is not under all walls. Should be attached the wall and columns to floor.



**Figure 19 The mistakes in building model**

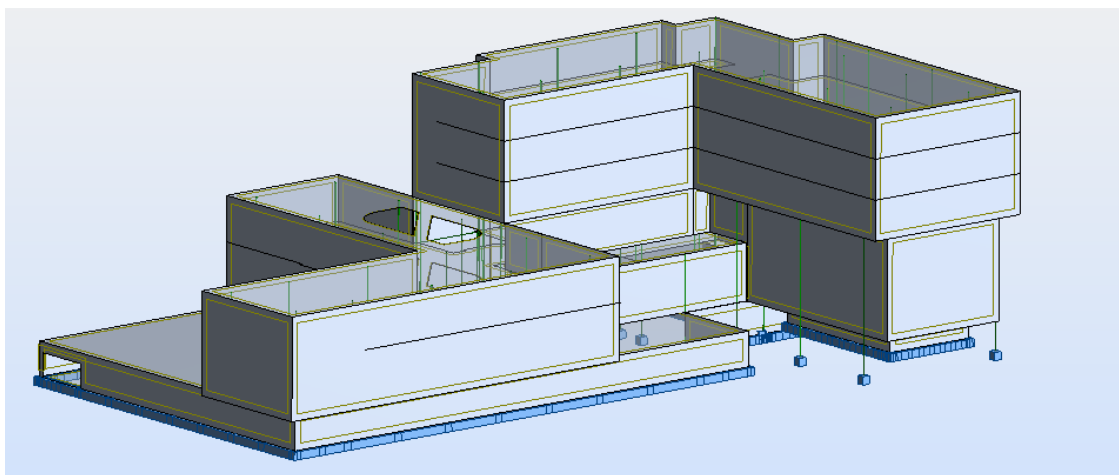


An important aspect is to check connections between the elements. The Revit program automatically between all the elements create pinned connection as shown in Figure 20. In this building model the releases have been between columns and beams, rest of the connections is rigid connection. In the case of reinforced concrete structure the best solution to chose is to make rigid connection for all elements.



**Figure 20** Releases

Due to the fact that main task of the outdoor walls is to protect the space from external factors that to carry the load, they are replaced by claddings. The main task is to transfer loads from the wind. As shown at Figure 21 claddings are not exactly located in the same places as the walls. Balconies were omitted in order to simplify the body of the building and for easier load distribution from the wind. Also, the main aim of this work is to show the possibility of cooperation between programs.



**Figure 21**Cladding

In the case of the cladding, it is possible to select the direction of the load distribution, it is possible distribution bi-directional, one direction x or y. Due to the different column grid, was adapted distribution in one direction. The load is spread on the ceilings of each floor. During

the creation the claddings, check the local system for these elements, because it can be different and, if necessary, change the weight distribution to a different direction as show in Figure 22.

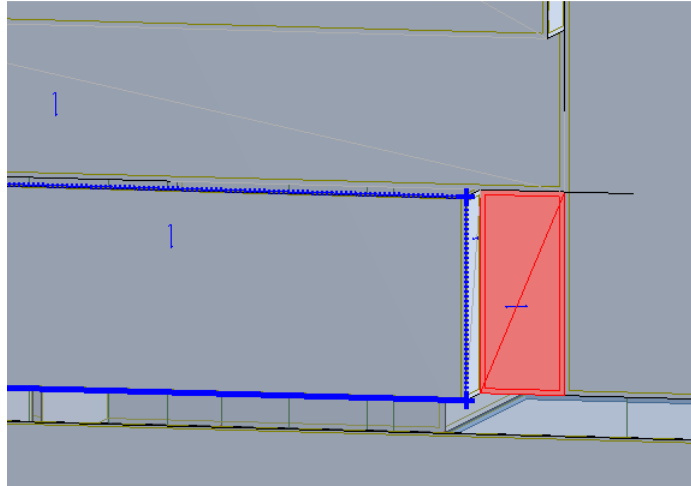


Figure 22 Load distribution

### 3.4 Apply case of loads

The next step is applying the various of the load which will be executed calculations. In the case of the designed building will be four different types of load. Dead weight is automatically generated from the program and takes account the weight of the structural elements. It is also important take into account the weight of its own layers of finishes and walls us the non-structural nature. Because of the building which is Hotel with Spa, it is necessary to add live load having regard to the different purpose of the rooms, values should be adopted based on norms. For the building should take into account the external load. The value of the load depends on the location and dimension. As shown at Figure 23 all loads are automatically generated in Revit, it is possible to change them, add extra or delete some of them. In case of Hotel with Spa all of them are not necessary, they will be removed. Applying an appropriate load is an important step in the design. On the basis of these calculations they will be made and selected the appropriate structural elements. In case of wrong calculation when in use may cause damage to some elements.

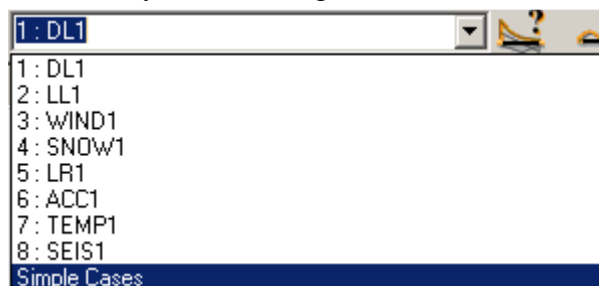


Figure 23 All loads from Revit

### 3.4.1 DL1 and DL2

The first one is self-weight DL1 which includes whole structures, columns, beams and walls, in addition to this case, is attached the load from the layers of finish coating thickness.

Values were calculated on the basis of Annex A from norm PN-EN 1991-1-1:2002 and Polish norm PN-82/B-02000.

The building has various types of ceilings.

- Table 1 shows calculation for two floors in higher part of the building. Contains written out individual layers, their thickness and weight. Calculated value is applied to the floor in Robot.

**Table 1 List of the loads on floor 1**

	Layers	Thickness	Volume weight	Surface weight
	-	[m]	[kN/m <sup>3</sup> ]	[kN/m <sup>2</sup> ]
1	Flooring	0,05	21,0	1,05
2	Board of the screed	0,05	21,0	1,05
3	PE foil	0,005	-	-
4	Gypsum board	0,025	12,0	0,30
5	Drywalls	0,02	7,9	0,158
				$g_k = 2,56$

- Table 2 shows calculation for ceiling in part devoted to the Spa and common rooms.

**Table 2 List of the loads on floor 0**

	Layers	Thickness	Volume weight	Surface weight
	-	[m]	[kN/m <sup>3</sup> ]	[kN/m <sup>2</sup> ]
1	Flooring	0,05	21,0	1,05
2	Board of the screed	0,05	21,0	1,05
3	PE foil	0,005	-	-
4	Gypsum board	0,025	12,0	0,30
5	Suspending ceiling	0,35	0,15	0,05
6	Mineral wool	0,10	2,0	0,20
7	Drywalls	0,02	7,9	0,16
				$g_k = 2,81$

- Table 3 shows calculation for floor in the garage.

**Table 3 List of the loads for garage**

	Layers	Thickness	Volume weight	Surface weight
	-	[m]	[kN/m <sup>3</sup> ]	[kN/m <sup>2</sup> ]

1	Concrete screed	0,05	24	1,20
2	Separating layer	0,005	-	-
3	Isolation	0,03	1,3	0,04
4	Vapor barrier PVC	0,01	-	-
5	Separating layer	0,005	-	-
6	Isolation XPS	0,05	0,5	0,03
7	Drainage of gravel	0,1	16,0	1,30
8	The protective membrane	0,005	-	-
				$g_k = 2,86$

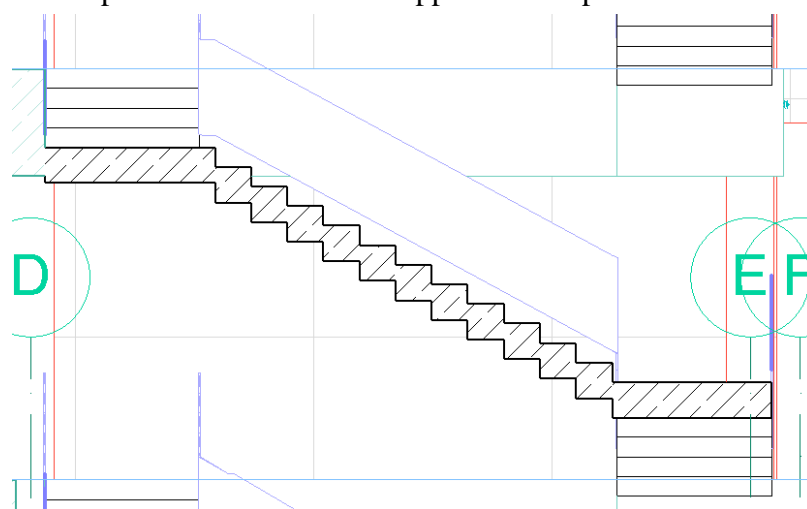
- Table 4 shows calculation for the roof.

**Table 4 List of the loads for roof**

	Layers	Thickness	Volume weight	Surface weight
	-	[m]	[kN/m <sup>3</sup> ]	[kN/m <sup>2</sup> ]
1	2x tar roofing	0,01	11	0,11
2	Isolation XPS	0,07	-	-
3	Vapor barrier PVC	0,005	-	-
4	Concrete screed	0,05	24	1,20
5	Drywalls	0,015	7,9	0,12
				$g_k = 1,43$

- Due to the fact that in the building model the stairs are omitted, they will be replaced with linear load. The building has four different kind of the stair. It will be calculated all their weight.

Figure 24 shows the stairs located in the higher part of the building. The stairs are located on all floors and will be replaced with linear load applied at the point of attachment.



**Figure 24 Calculated stairs nr 1**

The number of steps - 29

$h = 30 \text{ cm}$  - height of the stairs

$s = 30 \text{ cm}$  - width of the stairs

The number of landing - 2

$a = 130 \text{ cm}$  - length

$b = 160 \text{ cm}$  - width

Load from the stairs made of concrete

$$g_k = 25 \frac{\text{kN}}{\text{m}^3}$$

- weight of concrete

$$G_k = \frac{0,3 * 0,3 * 25 * 29 + 1,6 * 0,3 * 25 * 2}{2} = 44,63 \text{ kN/m}$$

- linear load applied along the length of 1,3m for both storey

Figure 25 shows the stairs located in the higher part of the building. The stairs are located on all floors and will be replaced with linear load applied at the point of attachment.

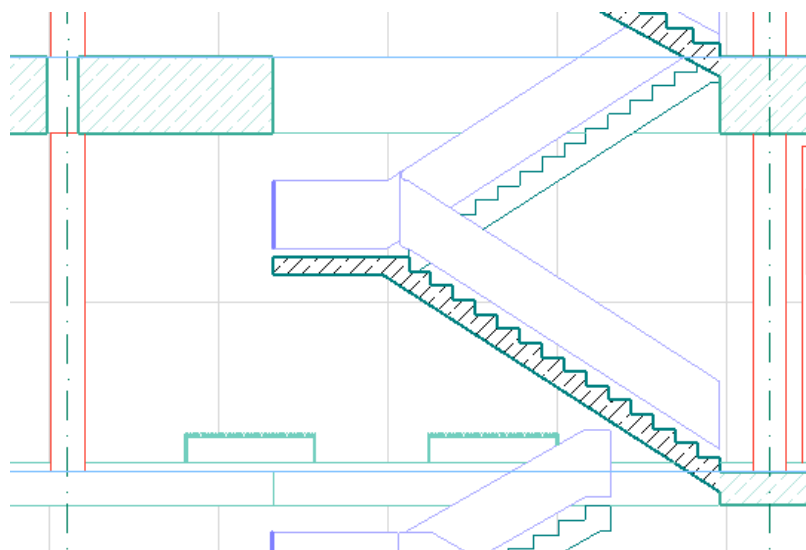


Figure 25 Calculated stairs nr 2

The number of steps - 20

$h = 17 \text{ cm}$  - height of the stairs

$s = 26 \text{ cm}$  - width of the stairs

$$\text{tg}\alpha = \frac{17}{26} = 0,654$$

$$\alpha = 33,179$$

$$\text{cos}\alpha = 0,837$$

The number of landing - 1

$a = 290 \text{ cm}$  - length

$b = 160 \text{ cm}$  - width

$c = 20 \text{ cm}$  - thickness

Load from the stairs made of concrete

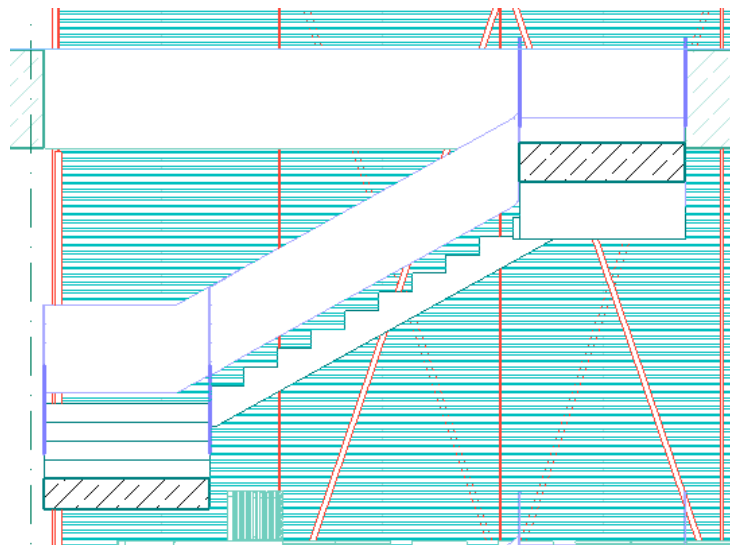
$$g_k = 25 \frac{\text{kN}}{\text{m}^3}$$

- weight of concrete ( the weight of the stair treads, flight of stairs, landing)

$$G_k = \frac{0,17 * 0,26 * 20/2 + 2 * 0,2 * 25/0,837 + 0,2 * 1,6 * 25}{2} = 10,19 \text{ kN/m}$$

- linear load applied along the length of 1,4m for both storey

Figure 26 shows the stairs located in the lower part of the building. The stairs are located on all floors and will be replaced with linear load applied at the point of attachment.



**Figure 26 Calculated stairs nr 3**

The number of steps - 27

$h = 16 \text{ cm}$  - height of the stairs

$s = 30 \text{ cm}$  - width of the stairs

$$\text{tg}\alpha = \frac{16}{30} = 0,533$$

$$\alpha = 28,07$$

$$\text{cos}\alpha = 0,882$$

The number of landing - 2

$a = 160 \text{ cm}$  - length

$b = 150 \text{ cm}$  - width

$c = 20 \text{ cm}$  - thickness

Load from the stairs made of concrete

$$g_k = 25 \frac{\text{kN}}{\text{m}^3}$$

- weight of concrete ( the weight of the stair treads, flight of stairs, landing)

$$G_k = \frac{0,16 * 0,30 * 27/2 + 3 * 0,2 * 25/0,882 + 2 * 0,2 * 1,6 * 25}{2} = 16,82 \text{ kN/m}$$

- linear load applied along the length of 1,5m for both storey

Figure 27 shows the stairs located in the lower part of the building. The stairs are located on all floors and will be replaced with linear load applied at the point of attachment.

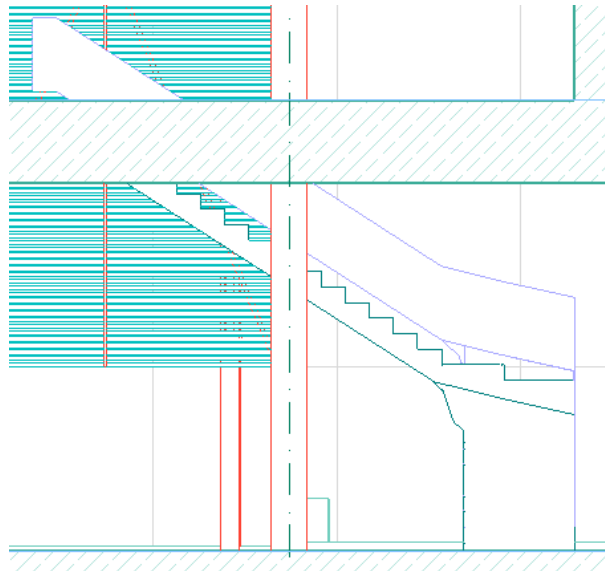


Figure 27 Calculated stair nr 4

The number of steps - 26

$h = 17 \text{ cm}$  - height of the stairs

$s = 26 \text{ cm}$  - width of the stairs

$$\text{tg}\alpha = \frac{17}{26} = 0,654$$

$\alpha = 33,179$

$\text{cos}\alpha = 0,837$

The number of landing - 2

$a = 120 \text{ cm}$  - length

$b = 120 \text{ cm}$  - width

$c = 20 \text{ cm}$  - thickness

Load from the stairs made of concrete

$$g_k = 25 \frac{\text{kN}}{\text{m}^3}$$

- weight of concrete ( the weight of the stair treads, flight of stairs, landing)

$$G_k = \frac{0,16 * 0,30 * 27/2 + 3 * 0,2 * 25/0,837 + 0,2 * 1,2 * 25}{2} = 15,25 \text{ kN/m}$$

- linear load applied along the length of 1,2m for both storey

### 3.4.2 LL1

The next case of load is operational load. The value of the load is taken on the basis of the norm PN-EN1991-1. In the building I have to different categories. For the bedroom at the hotel category A, for another rooms C3. (Table 6.1 and 6.2)

For the garage I have category F. ( Table 6.7 and 6.8)

For the loads I add the weight of the partition walls  $\rightarrow 1,2 \frac{kN}{m^2}$

$$g_{k1} = 2 \frac{kN}{m^2} + 1,2 \frac{kN}{m^2} = 3,2 \frac{kN}{m^2} \rightarrow \text{Category A – load for floor}$$

$$g_{k2} = 2 \frac{kN}{m^2} \rightarrow \text{Category A – load for stairs}$$

The value of live load acting on the stairs is converted to a linear load and applied at the connection between the stairs and the floor.

$$g_{k3} = 2,5 \frac{kN}{m^2} \rightarrow \text{Category A – load for balcony}$$

$$g_{k4} = 5 \frac{kN}{m^2} + 1,2 \frac{kN}{m^2} = 6,2 \frac{kN}{m^2} \rightarrow \text{Category C3}$$

$$g_{k5} = 2,5 \frac{kN}{m^2} \rightarrow \text{Category F}$$

### 3.4.3 WIND1

The next load is the external load from wind. The program Autodesk Robot Structural Analysis Professional 2015 gives the ability to generate automatic load from the wind. As show the Figure 28 to generate the wind, select the Loads ribbon, next Wind loads simulation and Generate wind loads. In the case of manual calculations had to divide the building into two separate and for them to perform the load from the wind. Ability to perform simulation in program makes the work easier and gives the opportunity to see the visualization of the load.



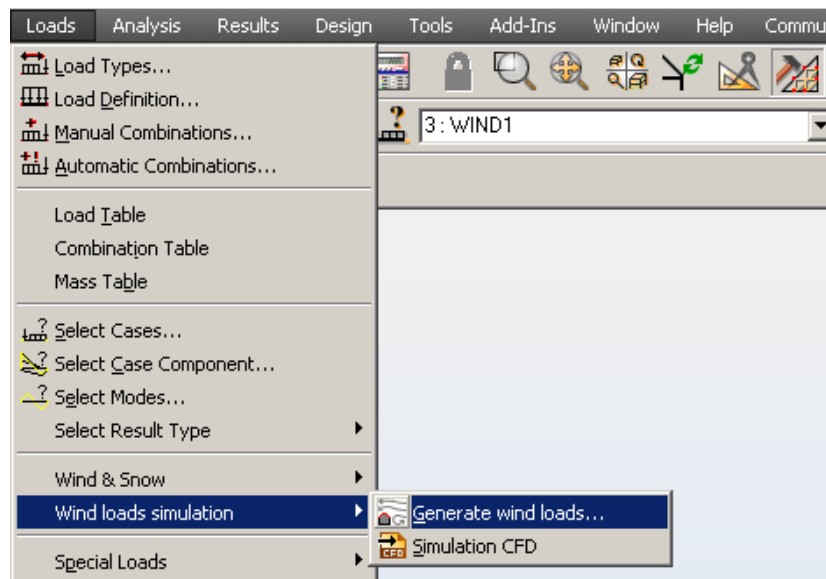


Figure 28 Generate wind loads

To generate the wind load it is necessary to calculate the value of wind velocity based on norm PN-EN 1991-1-4. In Poland there are three wind zones and information about the values are given in the National Annex.

Location of the building: Rzeszów → 1 wind load zone

Height above sea level for the Rzeszów:  $A = 206,75 \text{ m n.p.m.} < 300 \text{ m n.p.m.}$

Tablica NB.1. Wartości strefowe podstawowej wartości bazowej prędkości i ciśnienia prędkości wiatru;  
a – wysokość nad poziomem morza, m

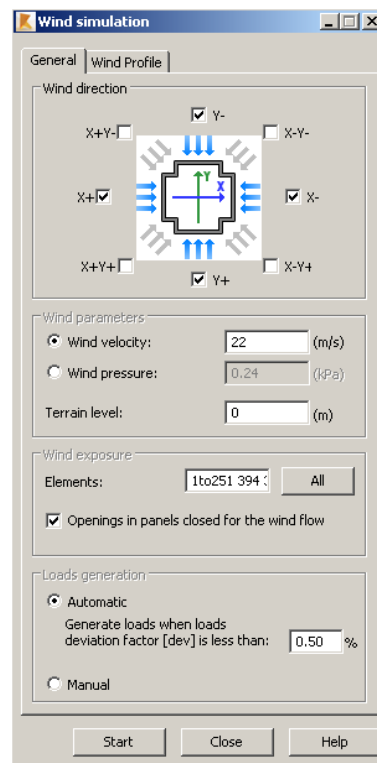
Strefa	$v_{b,0}$ , m/s	$v_{b,0}$ , m/s	$q_{b,0}$ , kN/m <sup>2</sup>	$q_{b,0}$ , kN/m <sup>2</sup>
	$a \leq 300 \text{ m}$	$a > 300 \text{ m}$	$a \leq 300 \text{ m}$	$a > 300 \text{ m}$
1	22	$22 \cdot [1 + 0,0006(a - 300)]$	0,30	$0,30 \cdot [1 + 0,0006(a - 300)]^2$
2	26	26	0,42	0,42
3	22	$22 \cdot [1 + 0,0006(a - 300)]$	0,30	$0,30 \cdot [1 + 0,0006(a - 300)]^2 \cdot \left[ \frac{20000 - a}{20000 + a} \right]$

Figure 29 Value of wind velocity

Alluding to table NB.1. from the norm:

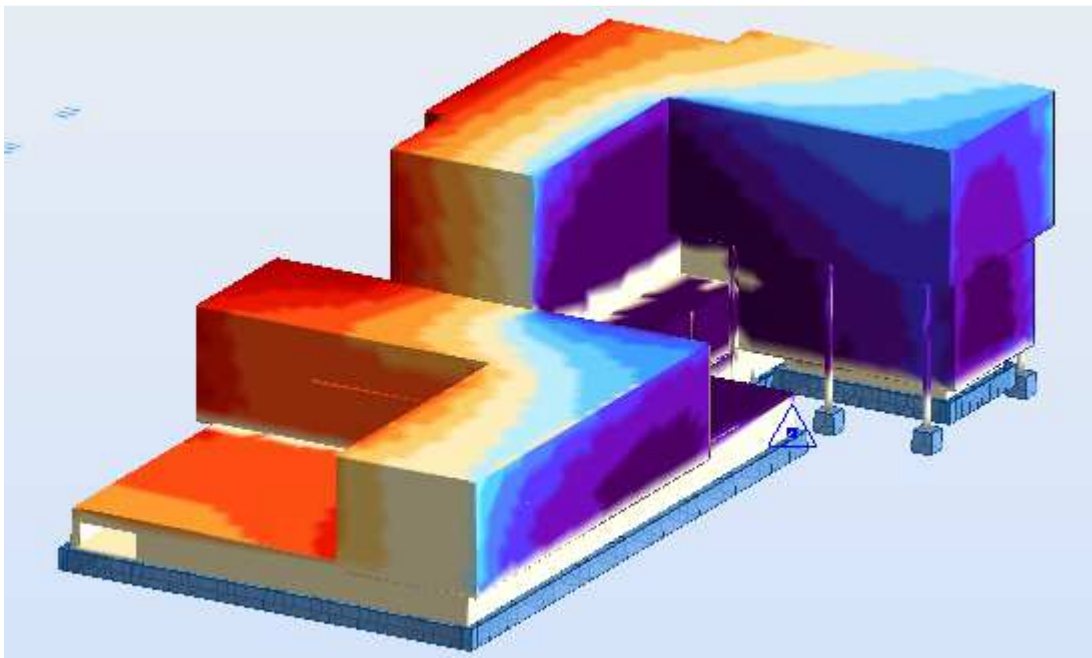
$$v_{b,0} = 22 \text{ m/s,}$$

As shown Figure 30 it is selected four different case of wind load, due to lack of symmetry in the object and different heights, in order to find the most disadvantageous. Due to the fact that the garage is located below ground level and it does not affect them loads of wind the terrain level is changed for 0 m. Then the next step is to press Start.



**Figure 30 Wind simulation**

Figure 31 shows the visualization of the wind in one direction. By colours there are identified different values of wind. The walls of the garage are in white colour because they have been omitted for these load.



**Figure 31 Visualization of wind**

As show Figure 32 as a result of simulation receive four additional cases of load. The loads are applied in accordance with the global system. In further actions have to be taken into account that at a time can affect the wind in one direction.

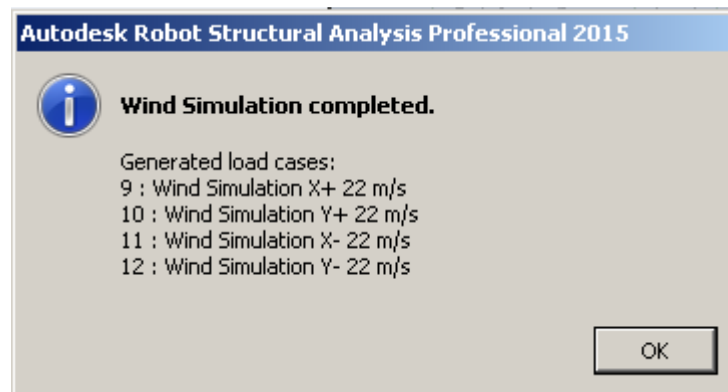


Figure 32 Information of completed the simulation

### 3.4.4 SNOW1

The last load is a load of snow applied to the roof of the building. Its value depends on the location. As show Figure 27 Poland has five zones of snow loads, depending on the height of the place above sea level.

Based on norm PN-EN 1991-1-3

Tablica NB.1 – Wartości charakterystyczne obciążenia śniegiem gruntu w Polsce

Strefa	$s_k, \text{kN/m}^2$
1	$0,007A - 1,4; \quad s_k \geq 0,70$
2	0,9
3	$0,006A - 0,6; \quad s_k \geq 1,2$
4	1,6
5	$0,93 \exp(0,00134A); \quad s_k \geq 2,0$

UWAGA: A = Wysokość nad poziomem morza (m)

Figure 33 Value of snow load

- Roof snow load for the design situation of permanent and transient

$$s = \mu_i \cdot C_e \cdot C_t \cdot s_k$$

- Coefficient the shape of the roof - flat roof

$$\mu_i = 0,8$$

- Characteristic value of snow load on the ground

$$s_k = 0,006A - 0,6;$$

$$s_k \geq 1,2 \text{ kN/m}^2$$

$$A = 206,75 \text{ m n.p.m}$$

$$s_k = 0,006 \cdot 206,75 - 0,6 = 0,641 \text{ kN/m}^2 < 1,2 \text{ kN/m}^2$$

Accepted

$$s_k = 1,2 \text{ kN/m}^2$$

- Expose index

$$C_e = 1,0 \rightarrow \text{for normal terrain}$$

- Thermal coefficient

$$C_t = 1,0$$

$$s = 0,8 \cdot 1,0 \cdot 1,0 \cdot 1,2 = 0,96 \text{ kN/m}^2$$

The value applied evenly to the roof is  $0,96 \text{ kN/m}^2$ .

After the application of all loads and generated the load of wind as shown Figure 34, there is eight case of load with different nature.

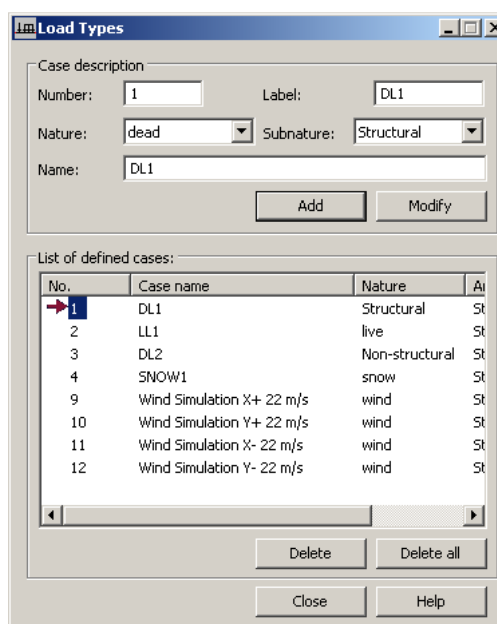
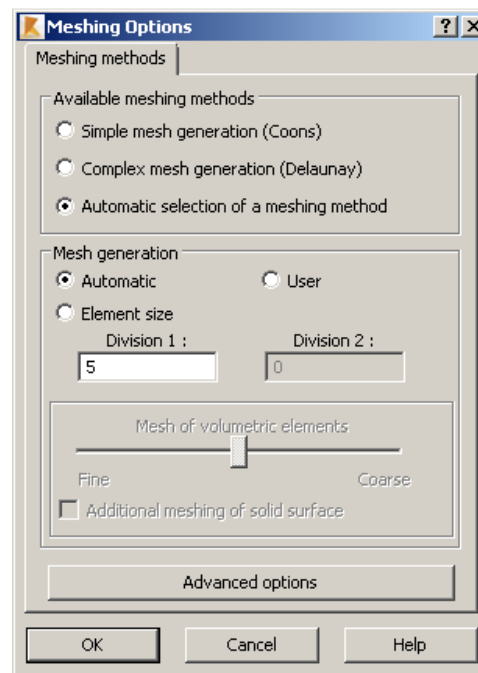


Figure 34 All cases of loads

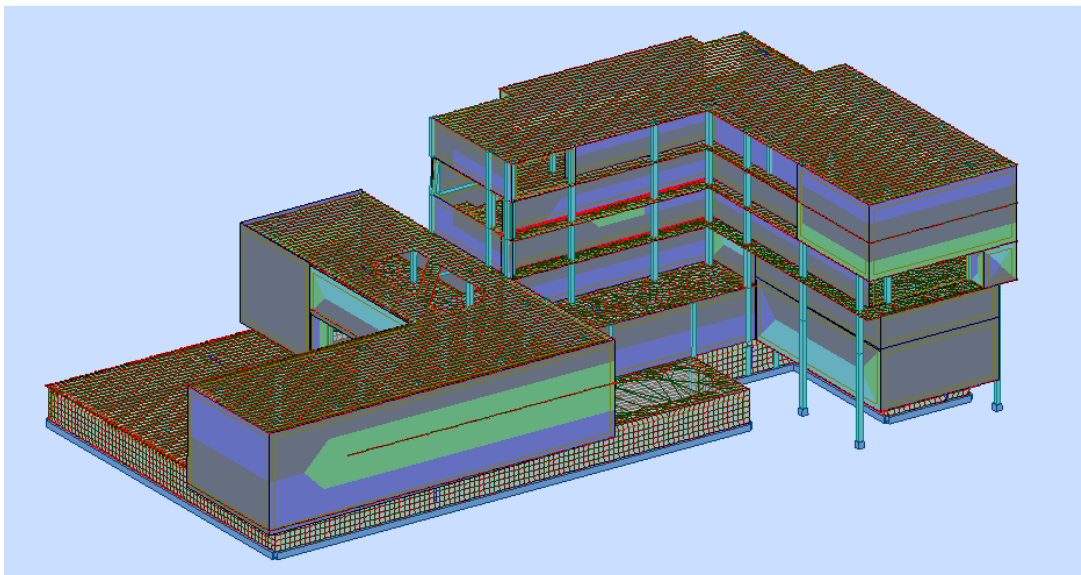
### 3.5 Generation of calculation model

The last stage before the calculation is a mesh generation. It was selected automatic selection of a meshing method as show Figure 35. Selection of appropriate methods and density of the mesh has an influence on later received results and their accuracy.



**Figure 35 Options of Meshing**

As show Figure 36 for the cladding is not generated mesh due to the fact that they are not structural elements.



**Figure 36 The building model with the mesh**

### 3.6 Combinations

For calculating the design should be taken into account the ultimate limit state SGN and serviceability limit state SGU based on norm PN-EN 1990.

Ultimate limit state concerns the safety of the people and structures.

Serviceability limit state applies the function structure or structural element under normal conditions of use, comfort of building occupants and appearance.

In the case of performing calculations in the Autodesk Robot Structural Analysis Professional can be made Manual Combinations and Automatic Combinations. In case of Manual Combinations it is necessary to create all combinations by oneself with regard to the equivalent coefficients.

In Automatic Combinations there is the possibility of full automatic combinations, simplified automatic combinations and manual combinations. Where could be selected different calculated cases, groups and creating relations between them.

In the case of the Hotel with Spa will be done manual combinations.

#### 3.6.1 SGN

Based on the Table 5, the value of the coefficients of the increasing will be taken account including whether the load is favorable or unfavorable. The coefficients also depends on whether the load is constant or variable.

Table 6 shows the value of the reducing factor depending on the load case and if it is the ultimate limit state or serviceability limit state.

**Table 5 The design value of impacts**

**Tablica A1.2(A) – Wartości obliczeniowe oddziaływań (EQU) (zestaw A)**

Trwale i przejściowe sytuacje obliczeniowe	Oddziaływania stałe		Wiodące oddziaływanie zmienne (*)	Towarzyszące oddziaływania zmienne	
	niekorzystne	korzystne		główne (jeżeli takie występują)	pozostałe
(Wzór 6.10)	$\gamma_{G,sup} \cdot Q_{k,sup}$	$\gamma_{G,int} \cdot Q_{k,int}$	$\gamma_{Q,1} \cdot Q_{k,1}$		$\gamma_{Q,i} \cdot \psi_{0,i} \cdot Q_{k,i}$
(*) Oddziaływaniami zmiennymi są te, które uwzględniono w tablicy A1.1  UWAGA 1 Wartości $\gamma$ mogą być podane w załączniku krajowym. Wartości zalecane podano poniżej: $\gamma_{G,sup} = 1,10$ $\gamma_{G,int} = 0,90$ $\gamma_{Q,1} = 1,50$ jeżeli niekorzystne (0 jeżeli korzystne) $\gamma_{Q,i} = 1,50$ jeżeli niekorzystne (0 jeżeli korzystne)  UWAGA 2 W przypadku, kiedy sprawdzenie równowagi statycznej uwzględnić także nośność elementów konstrukcji, można – jeżeli pozwala na to załącznik krajowy – zamiast dwukrotnego sprawdzenia korzystając z tablicy A1.2(A) i A1.2(B) – dokonać sprawdzenia jednokrotnego, posługując się tablicą A1.2(A) z podanym niżej zestawem wartości zalecanych. $\gamma_{G,sup} = 1,35$ $\gamma_{G,int} = 1,15$ $\gamma_{Q,1} = 1,50$ jeżeli niekorzystne (0 jeżeli korzystne) $\gamma_{Q,i} = 1,50$ jeżeli niekorzystne (0 jeżeli korzystne) Wartości zalecane mogą być zmienione w załączniku krajowym pod warunkiem, że przyjmując $\gamma_{G,int} = 1,00$ tak dla korzystnej jak i niekorzystnej części oddziaływań stałych, nie otrzymuje się mniej niekorzystnego efektu.					

**Table 6 The value of the coefficients**

**Tablica A 1.1 – Zalecane wartości współczynników  $\psi$  dla budynków**

Oddziaływania	$\psi_0$	$\psi_1$	$\psi_2$
Obciążenie zmienne w budynkach, kategoria (patrz EN 1991-1-1)			
Kategoria A: powierzchnie mieszkalne	0,7	0,5	0,3
Kategoria B: powierzchnie biurowe	0,7	0,5	0,3
Kategoria C: miejsca zebrań	0,7	0,7	0,6
Kategoria D: powierzchnie handlowe	0,7	0,7	0,6
Kategoria E: powierzchnie magazynowe	1,0	0,9	0,8
Kategoria F: powierzchnie ruchu pojazdów pojazdy $\leq 30$ kN	0,7	0,7	0,6
Kategoria G: powierzchnie ruchu pojazdów $30$ kN < ciężar pojazdu $\leq 160$ kN	0,7	0,5	0,3
Kategoria H: dachy	0	0	0,0
Obciążenie budynków śniegiem (patrz EN 1991-1-3) *)			
Finlandia, Islandia, Norwegia, Szwecja	0,70	0,50	0,20
Pozostałe kraje CEN, miejscowości położone na wysokości $H > 1000$ m ponad poziom morza	0,70	0,50	0,20
Pozostałe kraje CEN, miejscowości położone na wysokości $H \leq 1000$ m ponad poziom morza	0,50	0,20	0,20
Obciążenie wiatrem (patrz EN 1991-1-4)	0,6	0,2	0
Temperatura (nie pożarowa) w budynku (patrz EN 1991-1-5)	0,6	0,5	0
UWAGA: Wartości $\psi$ mogą być określone w załączniku krajowym *) Dotyczy krajów nie wymienionych poniżej – patrz miarodajne warunki miejscowe.			

Combination introduced to the program.

**Table 7 SNG Combinations**

COM 1	$1,35 \cdot (DL1 + DL2) + 1,5 \cdot 0,7 \cdot LL1 + 1,5 \cdot 0,5 \cdot SNOW1 + 1,5 \cdot 0,6 \cdot Wind X +$
COM 2	$1,35 \cdot (DL1 + DL2) + 1,5 \cdot 0,7 \cdot LL1 + 1,5 \cdot 0,5 \cdot SNOW1 + 1,5 \cdot 0,6 \cdot Wind Y +$
COM 3	$1,35 \cdot (DL1 + DL2) + 1,5 \cdot 0,7 \cdot LL1 + 1,5 \cdot 0,5 \cdot SNOW1 + 1,5 \cdot 0,6 \cdot Wind X -$
COM 4	$1,35 \cdot (DL1 + DL2) + 1,5 \cdot 0,7 \cdot LL1 + 1,5 \cdot 0,5 \cdot SNOW1 + 1,5 \cdot 0,6 \cdot Wind Y -$
COM 5	$1,35 \cdot 0,85 \cdot (DL1 + DL2) + 1,5 \cdot LL1 + 1,5 \cdot 0,5 \cdot SNOW1 + 1,5 \cdot 0,6 \cdot Wind X +$
COM 6	$1,35 \cdot 0,85 \cdot (DL1 + DL2) + 1,5 \cdot LL1 + 1,5 \cdot 0,5 \cdot SNOW1 + 1,5 \cdot 0,6 \cdot Wind Y +$
COM 7	$1,35 \cdot 0,85 \cdot (DL1 + DL2) + 1,5 \cdot LL1 + 1,5 \cdot 0,5 \cdot SNOW1 + 1,5 \cdot 0,6 \cdot Wind X -$
COM 8	$1,35 \cdot 0,85 \cdot (DL1 + DL2) + 1,5 \cdot LL1 + 1,5 \cdot 0,5 \cdot SNOW1 + 1,5 \cdot 0,6 \cdot Wind Y -$
COM 9	$1,35 \cdot 0,85 \cdot (DL1 + DL2) + 1,5 \cdot 0,7 \cdot LL1 + 1,5 \cdot SNOW1 + 1,5 \cdot 0,6 \cdot Wind X +$
COM 10	$1,35 \cdot 0,85 \cdot (DL1 + DL2) + 1,5 \cdot 0,7 \cdot LL1 + 1,5 \cdot SNOW1 + 1,5 \cdot 0,6 \cdot Wind Y +$

COM 11	$1,35 \cdot 0,85 \cdot (DL1 + DL2) + 1,5 \cdot 0,7 \cdot LL1 + 1,5 \cdot SNOW1 + 1,5 \cdot 0,6 \cdot Wind X -$
COM 12	$1,35 \cdot 0,85 \cdot (DL1 + DL2) + 1,5 \cdot 0,7 \cdot LL1 + 1,5 \cdot SNOW1 + 1,5 \cdot 0,6 \cdot Wind Y -$
COM 13	$1,35 \cdot 0,85 \cdot (DL1 + DL2) + 1,5 \cdot 0,7 \cdot LL1 + 1,5 \cdot 0,5 \cdot SNOW1 + 1,5 \cdot Wind X +$
COM 14	$1,35 \cdot 0,85 \cdot (DL1 + DL2) + 1,5 \cdot 0,7 \cdot LL1 + 1,5 \cdot 0,5 \cdot SNOW1 + 1,5 \cdot Wind Y +$
COM 15	$1,35 \cdot 0,85 \cdot (DL1 + DL2) + 1,5 \cdot 0,7 \cdot LL1 + 1,5 \cdot 0,5 \cdot SNOW1 + 1,5 \cdot Wind X -$
COM 16	$1,35 \cdot 0,85 \cdot (DL1 + DL2) + 1,5 \cdot 0,7 \cdot LL1 + 1,5 \cdot 0,5 \cdot SNOW1 + 1,5 \cdot Wind Y -$

### 3.6.2 SGU

COM 17:  $1,00 \cdot (DL1 + DL2) + 0,6 \cdot LL1 + 0,2 \cdot SNOW1$

### 3.7 Calculations

The next step is to perform the calculations in Robot. Based on the results will be possible dimensioning structural elements.

In Job preferences in the material sections was selected for all elements of concrete class C30/37.

Dimensioning of elements will be made on the basis of norm PN-EN 1992-1-1 and PN-EN 1992-1-2.

Bases on Table 4.1 from 1992-1 for columns will be selected the class designation - XC1, it is for elements inside building. For walls in garage and the columns outside the building it will be XF2 because of the possibility of expose to rain and freezing.

Structural Class: S4

Based on the Table 4.4N the environmental requirement for  $c_{min,dur}$  is 15mm for XC1 and

Based on The Regulation of the Minister of Infrastructure of 12 April 2002 from Poland. On the technical conditions to be met by buildings and their location. Dz.U.2015.0.1422

§8 Distribution of building for purposes of determining technical requirements and utility.

The height of the building is 19,50 m above ground level equivalent to medium-high (SW) - more than 12 m to 25 m.

Dz.U. Nr 56, poz. 461 z 2009r.

Pursuant to Article § 209 classified as the category of human risk ZL V.

Pursuant to Article § 212 based on the table required protection class - B



Pursuant to Article § 216 based on table fire resistance class for building elements.

Main supporting structure - R120

Roof construction - R30

### 3.7.1 Reinforcements for beams

The first element for which will be calculated reinforcement is beam. The building includes four beams around the holes for stairs. The dimensions of the elements is 40x50cm.

Based on Table 5.5 from PN-EN1992-1-2 standard fire resistance R 120 the axis is 55 mm.

- Concrete cover

$$c_{nom} = c_{min} + \Delta c_{dev}$$

$$c_{min} = \max \left\{ \begin{array}{l} c_{min,dur} + \Delta c_{dur,\gamma} - \Delta c_{dur,st} - \Delta c_{dur,add} \\ c_{min,b} \\ 10mm \end{array} \right.$$

$$c_{min,b} = 16 \text{ mm} - \text{adopted dimension of bar}$$

$$c_{min,dur} = 15 \text{ mm} - \text{based on Table 4.4 N}$$

$$\Delta c_{dur,\gamma} = \Delta c_{dur,st} = \Delta c_{dur,add} = 0$$

$$c_{min} = \max \left\{ \begin{array}{l} 16 \text{ mm} \\ 15 \text{ mm} \\ 10 \text{ mm} \end{array} \right. = 16 \text{ mm}$$

$$\Delta c_{dev} = 10 \text{ mm} - \text{value recommended by the norm}$$

$$c_{nom} = 16 + 10 = 26 \text{ mm}$$

The dimensions selected for stirrups 8 mm

$$a = c_{nom} + 8 \text{ mm} + \frac{\emptyset}{2} = 26 + 8 + \frac{16}{2} = 42 \text{ mm}$$

Due to the fire conditions the thickness of the cover is accepted 40 mm.

- The options selected in the program

The first element for which calculations are made regarding the bearing capacity and select the appropriate reinforcement, check whether the approved size is right for beams.

Performing calculation is possible when results in Robot are available.

In the Structure ribbon there was selected the beam.

First step is to select Provided Reinforcement of RC Elements. In new window choose the Manual combinations.

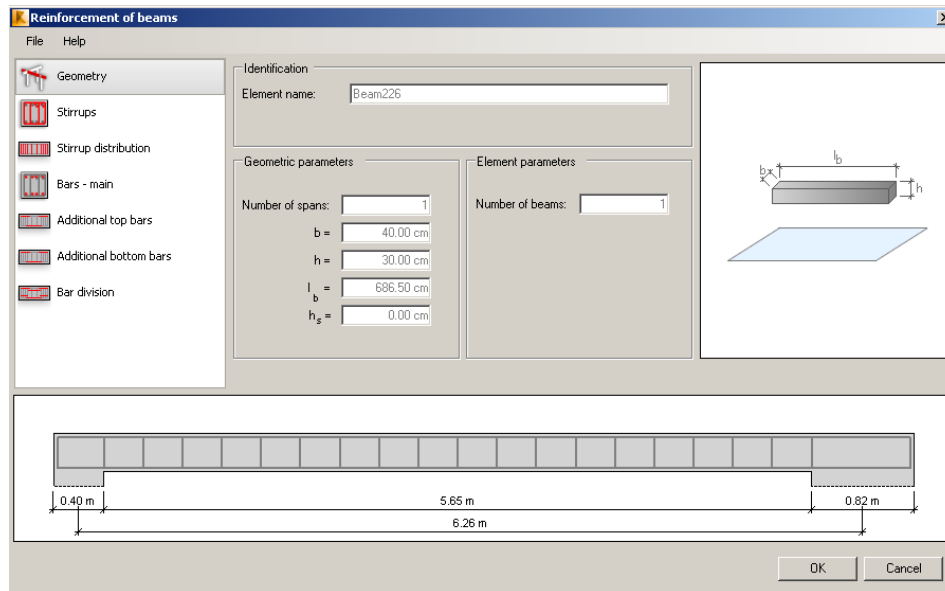
In Story Parameters the Environments class is XC1 and Structure class S4. Then the parameters are save as to select them for another elements.

In Calculation Options it has been included correction deflection by reinforcement change and  $w_{qp,li} < l_n/250$ . In Advanced Options was selected - Axial force taken into account and torsion taken into account. For fire provisions was selected the required fire resistance R120.

In next ribbon was selected concrete C30/37, for reinforcement AIIIIN (B500SP) and save as.

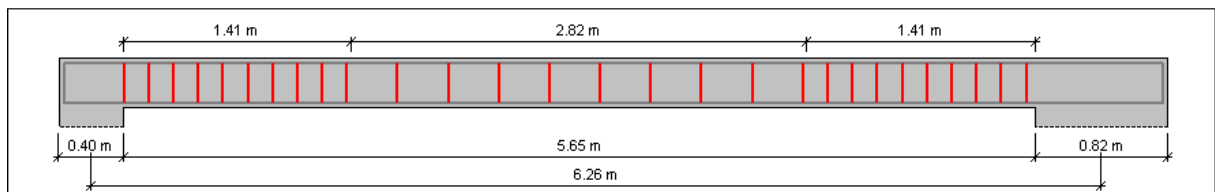
In options of reinforcement pattern minimum diameter of main reinforcement was selected 16 mm.

The last step in the Reinforcement of beams as show Figure 37.



**Figure 37 Reinforcement options**

For Stirrups the Bar size is 8 mm, Material A-IIIN (B500SP) and the cover 4 cm. The Stirrup distribution of main stirrups show the Figure 38.



**Figure 38 Distribution of main stirrups**

In Bars - main the size of Bar is 16 mm and Material A-IIIN (B500SP).

Initially was selected the number of lower bar - 4, upper bar - 2 and press OK to generate reinforcement and check whether it is appropriate.

Adopted properties are not sufficient to ensure adequate load bearing capacity of the beam.

In order to generate a suitable reinforcement there was selected calculations as show Figure 40 and generation of drawings.

No.	Type	State	Span	x [m]	Value	Capacity	Safety Fac.
<b>Beam223</b>							
1	M [kN*m]	ULS	1	6.05	-57.01	-9.39	0.16
2	VR [kN]	ULS	1	0.40	-138.20	0.00	-0.00
3	Abmin [cm <sup>2</sup> ]	SLS	1	0.40	2.62	0.48	0.18
4	Spacing [cm]	SLS	1	2.70	27.97	8.25	0.29
5	T [kN*m]	ULS	1	0.40	36.49	0.00	0.00
6	Cover does not satisfy requirements for the given environment						
7	Incorrect transverse spacing of the legs in a series of shear links (9.8N).						
8	Cover is too small in point 0.40(m)(3.3<5.2(cm))						

Figure 39 Calculation errors

Figure 40 Calculations

The program has generated reinforcement which satisfies pre-established parameters. It has increased the number of top reinforcement and added in the area of the supports as show the Figure 41. Correction of drawings will be made in the Revit program.

General   Detailed   Summary table   Spacings and areas										
No.	Reinforcement Type	Steel Grade	Diameter (mm)	Shape Code	Number	(m)	(m)	(m)	(m)	(m)
1	main-top	A-IIIIN (B500	12	00	4	A = 6.43				
2	main-bottom	A-IIIIN (B500	12	00	4	A = 6.25				
3	transverse-main	A-IIIIN (B500	8	31	27	A = 0.29	B = 0.41	C = 0.29	D = 0.41	
*										

Figure 41 Generated reinforcement

The same actions as the above must be applied to the remaining beams for the purpose of calculating.

After calculations on RC component inspector they are all elements with the information if the calculations are correct as show the Figure 42. It also allows for easy selection of elements and shows their position relative to the individual floors.

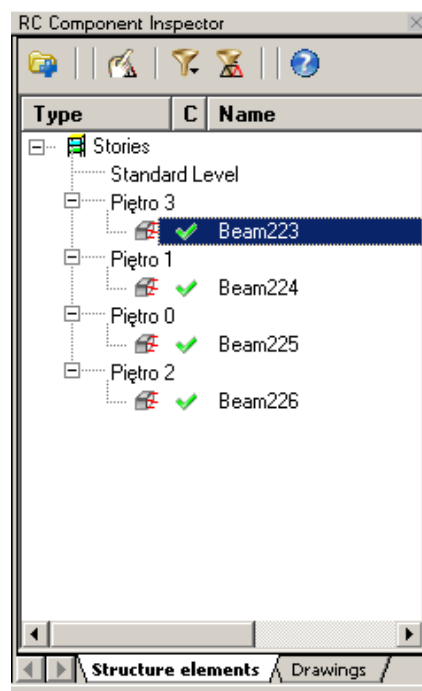


Figure 42 Calculated elements

### 3.7.2 Reinforcements for columns

The next elements to be calculated is columns.

Based on Table 5.2a from norm PN-EN 1992-1-2 for standard fire resistance R120 and columns exposed on more than one side the distance of the main bars would be 45 mm and minimum 8 bars.

- The options selected in the program

After selected one of the columns from last floor, was chosen Provided Reinforcement of RC Elements and the Manual combinations.

The Environment class XC1 and Structure class S4 in story parameters.

For fire provisions was selected R 120 and Method B. In next ribbon the class of concrete C30/37.

For reinforcement of columns the dimension of bars 16 mm and of stirrups 8 mm.

The dowels are not selected for last floor.

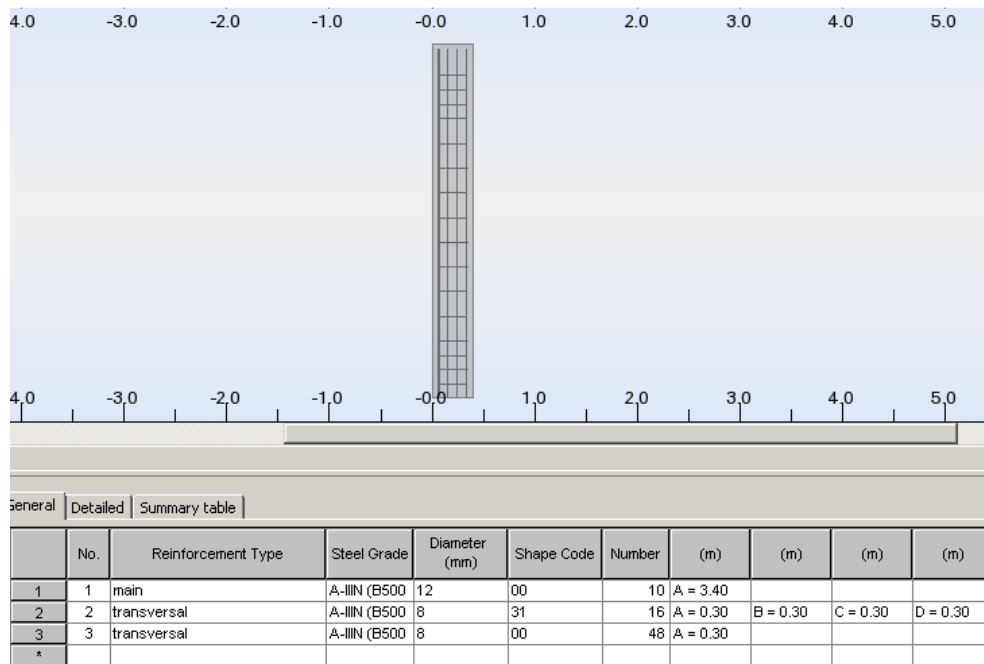


Figure 43 Calculated reinforcement

For all columns on the last floor the options are selected the same.

For lower floors in Dowel Bars ribbon the connections to the upper column and bend bars is selected On. After that it is possible to connect two columns as show Figure 44.

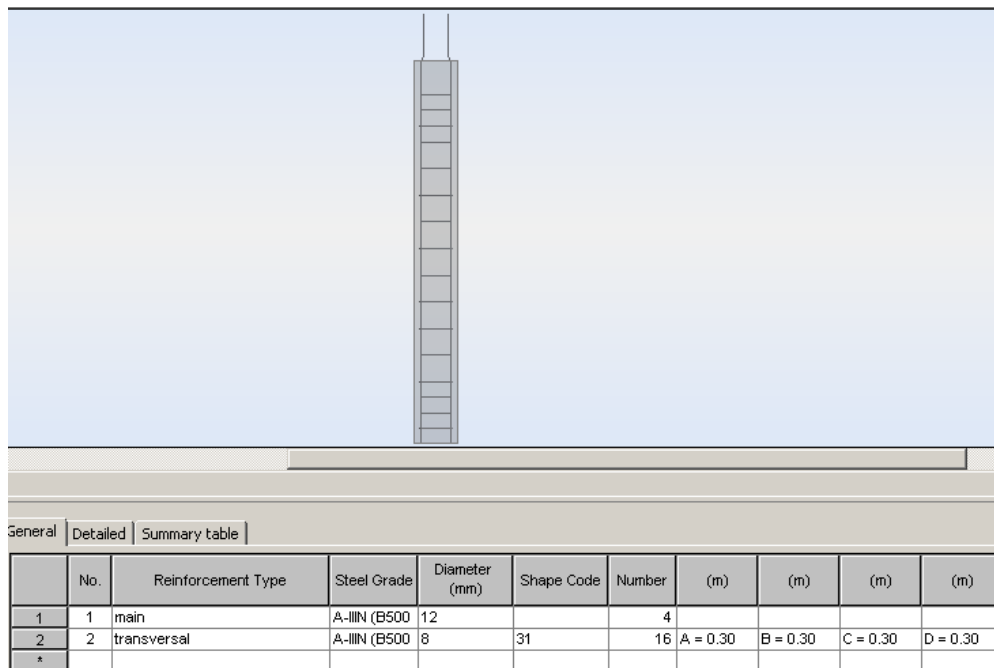


Figure 44 Calculated element for lower floor

For lower part of the building should be also skip the dower bars.

For the columns which are outside the building the environment class is change to XF2.

After performing the calculation to design the reinforcement, the results obtained are different depending on the forces what has to move the column. For purpose of convenience during the construction of the building for each floor reinforcement will be chosen the same. Also due to the fire conditions contained in the norm minimum number of bars of the main reinforcement for columns is 8. The drawings for reinforcement will be corrected in the Revit program.

### 3.7.3 Reinforcements for slabs

The first step to check the required reinforcement for slabs in the building.

Based on Table 5.8 for slabs and standard fire resistance REI 120 minimum axis distance 25 mm.

In Slab - required reinforcement was selected all panels in the model. They were selected properly combinations and the analytical method. It was enabled reduction of forces, this means that the value of the moments and stresses in the area of the supporting points is replaced by the mean value of the surrounding columns or supports. After calculation selected in scale ribbon 256 colours and in reinforcement ribbon Area A Y[-]. It is allows to see the results as show the Figure 45.

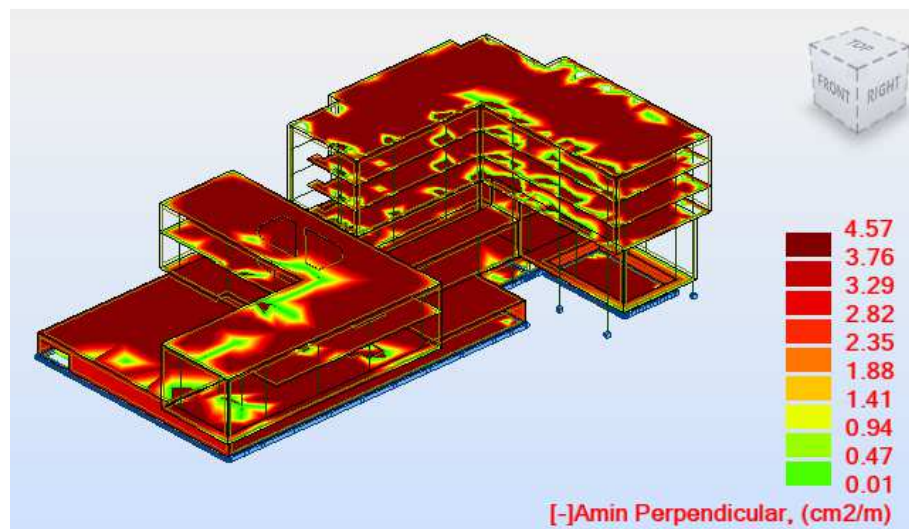
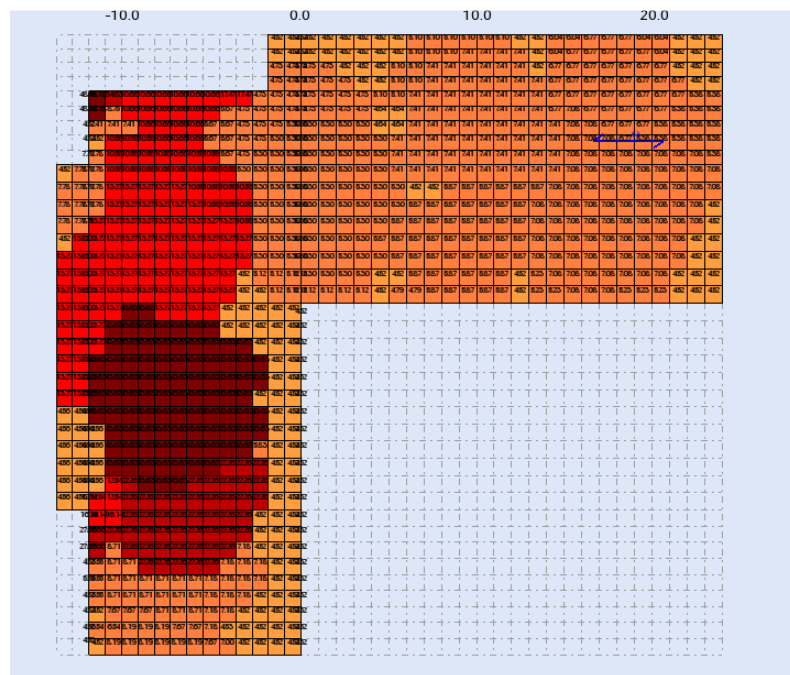


Figure 45 Required reinforcement

In order to calculate the provided reinforcement was selected the last slab.

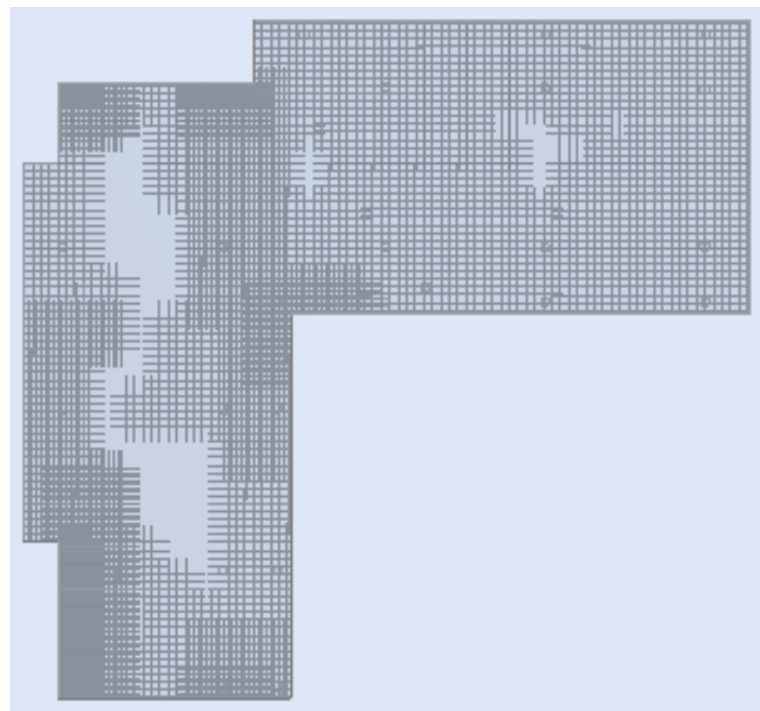
In the new window it is possible to see the whole slab with a variety of colours and the value of reinforcement in  $\text{cm}^2/\text{m}$  for selected direction of reinforcement as show Figure 46.

In order to design a provided reinforcement, it is important to establish appropriate parameters. In Reinforcement pattern was selected the bars type.

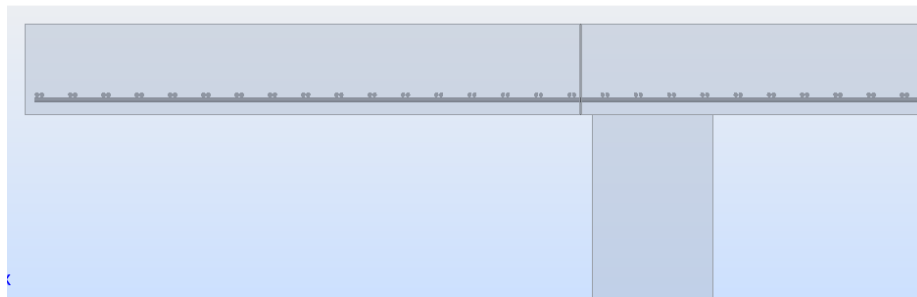


**Figure 46 Required reinforcement for last slab**

In order to made the accurately calculation it would be selected the Slab - reinforcement ribbon. In calculation options for Reinforcing bars was selected dimension from 16 to 32 mm. For story parameters was selected structural class S4 and pressed the button Calculations. Based on the applied load was calculated top and bottom reinforcement as show Figure 47 and 48.



**Figure 47 Top reinforcement for roof**

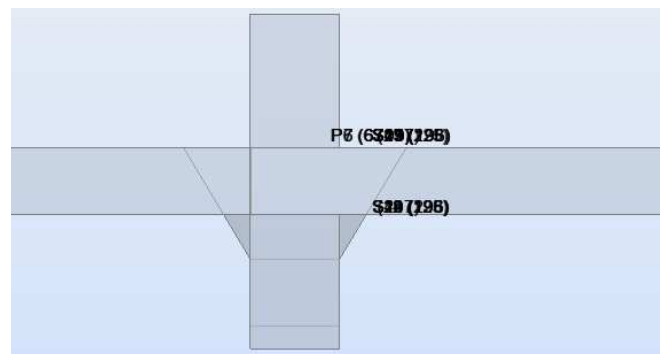


**Figure 48 Bottom reinforcement for roof**

The same operations would be performed in order to design reinforcement for the other slabs. Then all the drawings would be corrected in Revit program.

In case of the lower floor load bearing capacity of the floor is not sufficient, in case of design in BIM technology next step would be to contact with architect and together discuss a possible solution. One of the solution could be to increase the thickness of the floor, however, it will reduce the height of the room. Another possible solution is to reduce the space between columns, but it requires approval of the architect. Therefore, in order not to interfere with the appearance of the building will be increased concrete class for C45/55. After changing the concrete class for the structural elements must be repeated calculations.

The final step is to check the punching for slabs in Robot program. For places where the bearing capacity is insufficient, the punching force value is too big, would be design the head of rectangular shape as show Figure 49.



**Figure 49 Shell reinforcement**



### 3.7.4 Reinforcements for walls

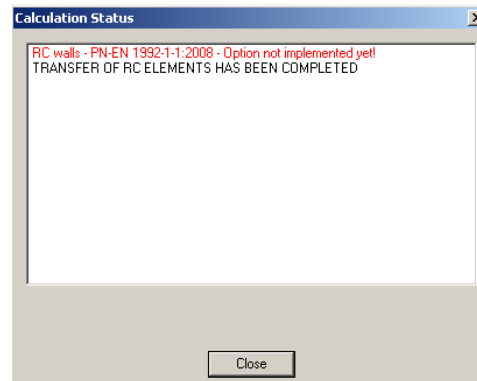


Figure 50 Calculation status for walls

As show the Figure 50 there is no possibility of calculating the needed reinforcement for the walls, they are made manual.

- Concrete cover

$$c_{nom} = c_{min} + \Delta c_{dev}$$

$$c_{min} = \max \left\{ \begin{array}{l} c_{min,b} \\ c_{min,dur} + \Delta c_{dur,\gamma} - \Delta c_{dur,st} - \Delta c_{dur,add} \\ 10mm \end{array} \right.$$

$$c_{min,b} = 16 \text{ mm} - \text{adopted dimension of bar}$$

$$c_{min,dur} = 15 \text{ mm} - \text{based on Table 4.4 N}$$

$$\Delta c_{dur,\gamma} = \Delta c_{dur,st} = \Delta c_{dur,add} = 0$$

$$c_{min} = \max \left\{ \begin{array}{l} 16 \text{ mm} \\ 15 \text{ mm} \\ 10 \text{ mm} \end{array} \right. = 16 \text{ mm}$$

$$\Delta c_{dev} = 10 \text{ mm} - \text{value recommended by the norm}$$

$$c_{nom} = 16 + 10 = 26 \text{ mm}$$

The dimensions selected for stirrups 8 mm

$$a = c_{nom} + 8 \text{ mm} + \frac{\emptyset}{2} = 26 + 8 + \frac{16}{2} = 42 \text{ mm}$$

Based on Table 5.4 for load-bearing concrete walls and standard fire resistance REI 120, minimum thickness 160 mm and the axis distance 35 mm in case of exposed on one side.

Due to the fire conditions the thickness of the cover is accepted 40 mm.

Based on the value from the calculation in the Robot program which are in one direction much larger assumes that the wall would be bending in one direction and is mainly compression.

- Material characteristics

#### Concrete C45/55

$$f_{ck} = 45 \text{ MPa}$$

$$f_{ctm} = 30 \text{ MPa}$$

$$f_{ctk,0,05} = 2,7 \text{ MPa}$$

$$\gamma_c = 1,4$$

$$f_{cd} = \frac{f_{ck}}{\gamma_c} = \frac{45}{1,4} = 32,14 \text{ MPa}$$

$$f_{ctd} = \frac{f_{ctk,0,05}}{\gamma_c} = \frac{2,7}{1,4} = 1,93 \text{ MPa}$$

$$\varepsilon_{cu2} = 3,5\text{‰}$$

### Steel B500SP

$$f_{yk} = 500 \text{ MPa}$$

$$\gamma_s = 1,15$$

$$f_{yd} = \frac{f_{yk}}{\gamma_s} = \frac{500}{1,15} = 435 \text{ MPa}$$

$$\varepsilon_{yd} = \frac{435}{200000} = 2,18\text{‰}$$

$h = 36 \text{ cm}$  - wall thickness

$b = 100 \text{ cm}$  - calculations were performed for the section of 1 meter wall

$$A_c = b \cdot h = 1,0 \cdot 0,36 = 0,36 \text{ m}$$

Moment of inertia

$$I = \frac{b \cdot h^3}{12} = \frac{1,0 \cdot 0,36^3}{12} = 3,89 * 10^{-3} \text{ m}^4$$

$$i = \left( \frac{I}{A_c} \right)^{0,5} = \left( \frac{3,89 * 10^{-3}}{0,36} \right)^{0,5} = 0,104 \text{ m}$$

$$a_1 = c_{nom} + \frac{\varnothing}{2} = 40 + \frac{16}{2} = 48 \text{ mm}$$

$$d = h - a_1 = 360 - 48 = 312 \text{ mm}$$

$$x_{lim} = \varepsilon_{cu2} / (\varepsilon_{cu2} + \varepsilon_{yd}) \cdot d = 3,5 / (3,5 + 2,18) \cdot 312 = 192,25 \text{ mm}$$

- Effective length and slenderness

$\beta = 0,85$  → based on the information from norm PN-EN 1992-1-1, 12.6.5.1 (4) a wall connected in flexurally rigid manner

$$l_w = 3,0 \text{ m} - \text{height of the wall}$$

$$l_0 = \beta \cdot l_w = 0,85 \cdot 3000 = 2550 \text{ mm}$$

$$\lambda = \frac{l_0}{i} = \frac{2,55}{0,104} = 24,54$$

- Geometric imperfections

$$\theta_i = \theta_0 \cdot \alpha_h \cdot \alpha_m \text{ (5.1) PN-EN 1992-1-1}$$

$$\theta_0 = \frac{1}{200}$$

$m = 1$  - the number of the elements

$$\alpha_m = \sqrt{0,5 \cdot \left(1 + \frac{1}{m}\right)} = \sqrt{0,5 \cdot (1 + 1)} = 1$$

$\alpha_h = 1$  – based on the information from the norm 5.2 (7)

$$\theta_i = \frac{1}{200} \cdot 1 \cdot 1 = 0,005$$

$e_i = l_0/400$  - information from the norm for walls

$$e_i = \frac{l_0}{400} = \frac{2550}{400} = 6,38 \text{ mm}$$

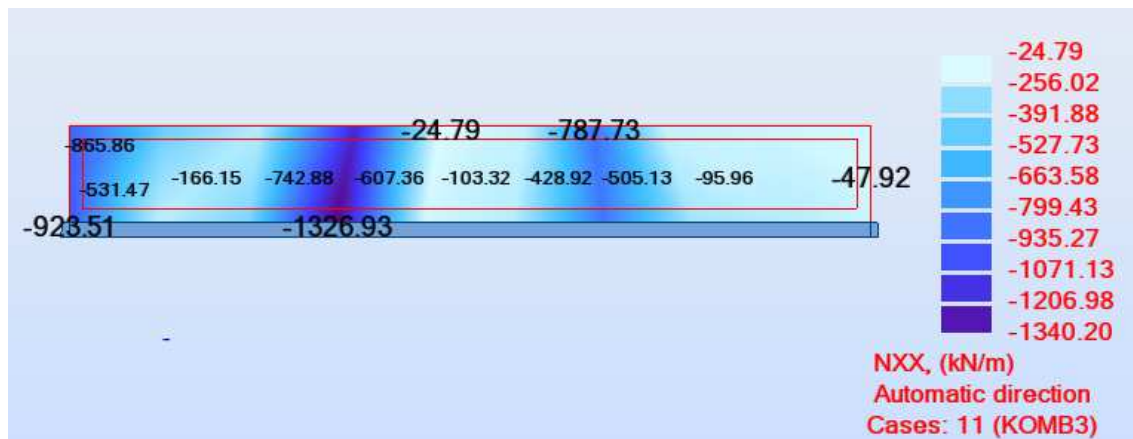


Figure 51 Member forces N in direction xx

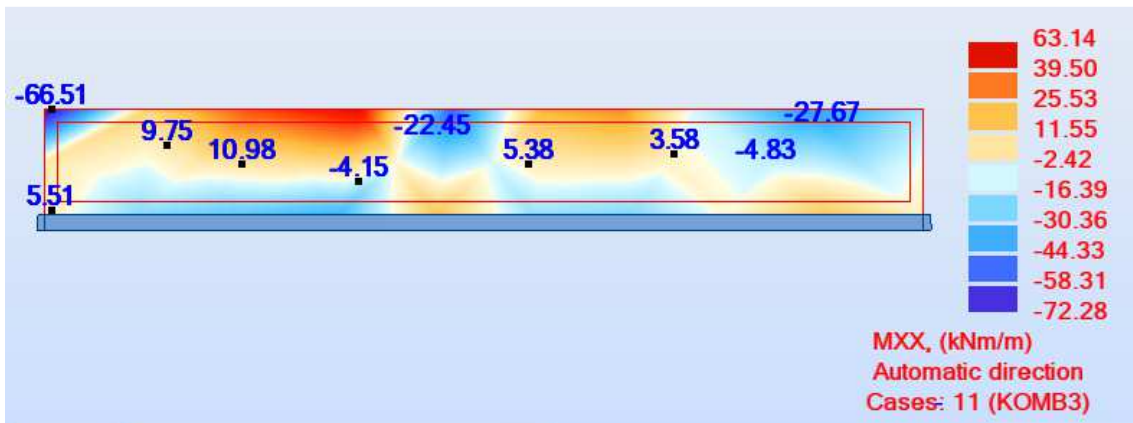


Figure 52 Moments in direction xx

Figure 51 and 52 shows the forces for one of the combinations which is the most disadvantageous. Based on that calculations from the Robot program would be calculated the necessary reinforcement.

$$N_{Ed} = 1340,20 \text{ kN}$$

$$M_{Ed} = 72,28 \text{ kNm}$$

$$M_{0Ed} = e_i \cdot N_{Ed} + M_{Ed} = 0,0064 \cdot 1340,2 + 72,28 = 80,82 \text{ kNm}$$

$$e_0 = \max \left\{ \begin{array}{l} \frac{M_{0Ed}}{N_{Ed}} \\ h/30 \\ 20 \text{ mm} \end{array} \right.$$

$$e_0 = \max \left\{ \begin{array}{l} \frac{80,82}{1340,2} = 0,060 \text{ m} = 60 \text{ mm} \\ \frac{360}{30} = 12 \text{ mm} \\ 20 \text{ mm} \end{array} \right.$$

$$e_0 = 60 \text{ mm}$$

$$M_{0Ed} = e_0 \cdot N_{Ed} + M_{Ed} = 0,060 \cdot 1340,2 + 72,28 = 153,10 \text{ kNm}$$

- Slenderness criterion

$$\lambda_{lim} = \frac{20 \cdot A \cdot B \cdot C}{\sqrt{n}}$$

According to 5.8.3.1 PN-EN 1992-1-1

$$A = 0,7$$

$$B = 1,1$$

$$C = 0,7$$

$$n = \frac{N_{Ed}}{A_c \cdot f_{cd}} = \frac{1340,2}{0,36 \cdot 32143} = 0,12$$

$$\lambda_{lim} = \frac{20 \cdot 0,7 \cdot 1,1 \cdot 0,7}{\sqrt{0,12}} = 31,68$$

$$\lambda = 24,54 < \lambda_{lim} = 31,68 \rightarrow \text{its not necessary to include the second order effects}$$

- Calculation of vertical reinforcement  
- the position of the neutral axis of sections

$$x = \frac{N_{Ed}}{0,8 \cdot b \cdot f_{cd}} = \frac{1340,2}{0,8 \cdot 1,0 \cdot 32143} = 0,0521 \text{ m} = 52,1 \text{ mm} < x_{lim} = 192,25 \text{ mm}$$

← case of large eccentricity

$$A_1 = A_2 = \frac{M_{0Ed} - 0,5 \cdot \eta \cdot f_{cd} \cdot b \cdot \lambda \cdot x \cdot (h - \lambda \cdot x)}{(d - a_1) \cdot f_{yd}}$$

$$\lambda = 0,8; \eta = 1,0 \text{ for class of concrete } \leq C50/60$$

$$A_1 = A_2 = \frac{0,153 - 0,5 \cdot 1,0 \cdot 32,14 \cdot 1,0 \cdot 0,8 \cdot 0,0539 \cdot (0,36 - 0,8 \cdot 0,0539)}{(0,312 - 0,048) \cdot 435} = -0,00052 \text{ m}^2 - \text{reinforcement due}$$

to bending is not required

- Minimum vertical reinforcement

$$A_{s,min} = \max \left\{ \begin{array}{l} \frac{0,10 \cdot N_{Ed}}{f_{yd}} \\ 0,002 \cdot A_c \end{array} \right.$$

$$A_{s,min} = \max \left\{ \begin{array}{l} \frac{0,10 \cdot 1,3840}{435} = 3,08 \cdot 10^{-4} m^2 \\ 0,002 \cdot 0,36 = 7,2 \cdot 10^{-4} m^2 \end{array} \right.$$

$s \leq 400 \text{ mm} - \text{bar spacing}$

$A_{s1,min} = 7,2 \cdot 10^{-4} m^2$  - the reinforcement required attributable to 1 meter of wall

$$A_1 = A_2 = 3,6 \cdot 10^{-4} m^2$$

Accepted reinforcement -  $A_1 = A_2 = 3,77 \text{ cm}^2$  -  $\emptyset 12$  every 30 cm

- Minimum horizontal reinforcement

$$A_{s2,min} = \max \left\{ \begin{array}{l} 0,25 \cdot A_{s1,min} \\ 0,001 \cdot A_c \end{array} \right.$$

$$A_{s2,min} = \max \left\{ \begin{array}{l} 0,25 \cdot 7,54 \cdot 10^{-4} = 1,89 \cdot 10^{-4} m^2 \\ 0,001 \cdot 0,36 = 3,6 \cdot 10^{-4} m^2 \end{array} \right.$$

$$A_{s2,min} = 3,6 \cdot 10^{-4} m^2$$

$s \leq 400 \text{ mm} - \text{bar spacing}$

Accepted reinforcement -  $A_1 = A_2 = 3,39 \text{ cm}^2$  -  $\emptyset 12$  every 40 cm

- Minimum transverse reinforcement

Based on the norm the transverse reinforcement is not required

$$A_{s1,min} = 7,54 \cdot 10^{-4} m^2 < 0,02 \cdot A_c = 72 \cdot 10^{-4} m^2$$

- Checking the ultimate limit state ( Starosolski, 2007)

$$N_{ed} \leq N_{Rd} = \Phi \cdot f_{cd} \cdot h_w \cdot b$$

$$h_w = 0,36 \text{ m}$$

$$b = 1,0 \text{ m}$$

$$l_{eff} = 1,3 l_0$$

$$l_0 = 3,0 \text{ m}$$

$$l_{eff} = 1,3 \cdot 3,0 = 3,9 \text{ m}$$

$$e_0 = e_e + e_a$$

$$e_e = \frac{M_{Ed}}{N_{Ed}} = \frac{72,28}{1340,2} = 0,05 \text{ m}$$

$$e_a = \max \left\{ \begin{array}{l} \frac{l_{col}}{600} \\ h \\ \frac{30}{10} \end{array} \right.$$

$$e_a = \max \begin{cases} \frac{3000}{600} = 5 \text{ mm} \\ \frac{36}{30} = 1,2 \text{ mm} \\ 10 \text{ mm} \end{cases}$$

$$e_a = 10 \text{ mm}$$

$$e_0 = e_e + e_a = 0,05 + 0,01 = 0,06 \text{ m}$$

$$\frac{l_{eff}}{h_w} = \frac{3,9}{0,36} = 11$$

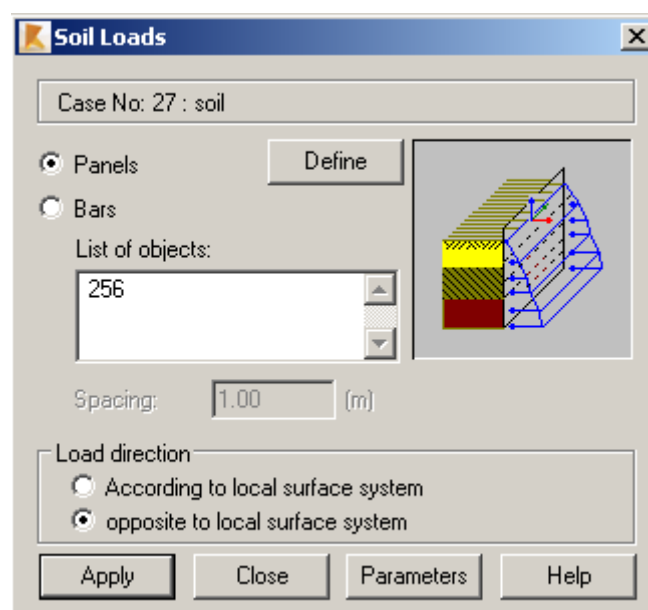
$$\frac{e_0}{h_w} = \frac{0,06}{0,36} = 0,20$$

$\Phi = 0,51$  - based on Tab. 7.2 (Starosolski, 2007)

$$N_{Rd} = 0,51 \cdot 32143 \cdot 0,36 \cdot 1,0 = 5901 \text{ kN}$$

$$N_{ed} = 1340,20 \text{ kN} \leq N_{Rd} = 5901 \text{ kN}$$

- Pressure from the soil



**Figure 53 Soil loads**

For soil load would be created new load case. To apply the load it is necessary to select the special loads and next soil pressure in Robot. As show Figure 53 next step is to change the parameters. There was selected the clayey gravel, the soil level 0,00 m and the category of pressure distribution uniform. Then was selected all exterior walls to apply the loads.

For combination the value of soil loads was multiply by 1,35. Based on the result would be calculated reinforcement and will be added to this which was calculated above.

- Section height of the useful

Accepted reinforcement  $\varnothing 12$  and concrete cover 40 mm.

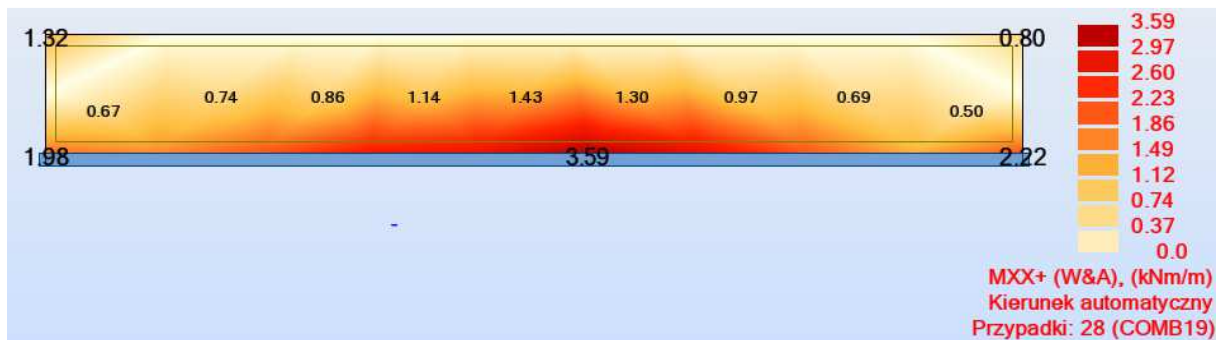


Figure 54 The moments value of the soil load

The next step is to check if the reinforcement selected above will be sufficient to move the load from the soil.

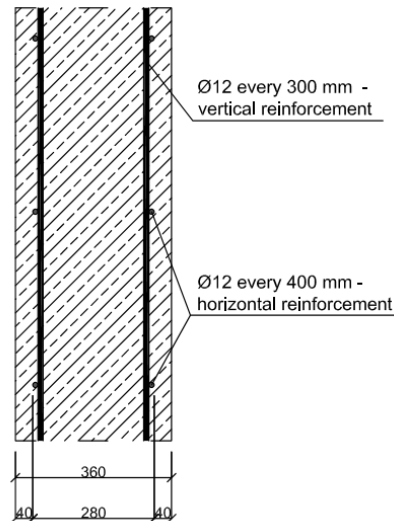


Figure 55 Scheme of reinforcement

$$d_v = h - c_{nom} - \frac{\varnothing}{2} - \varnothing = 360 - 40 - \frac{12}{2} - 12 = 302 \text{ mm}$$

$$A_1 = 3,77 \text{ cm}^2 - \text{reinforcement accepted earlier}$$

$$x_{eff} = \frac{A_1 \cdot f_{yd}}{1 \text{ m} \cdot f_{cd}} = \frac{3,77 \cdot 10^{-4} \cdot 435}{1 \cdot 32,14} = 0,005 \text{ m}$$

$$M_{Rd} = A_1 \cdot f_{yd} \cdot \left( d_v - \frac{x_{eff}}{2} \right) = 3,77 \cdot 10^{-4} \cdot 435 \cdot \left( 0,302 - \frac{0,005}{2} \right) = 0,049 = 49 \text{ kNm}$$

$M_{Rd} = 49 \text{ kNm} > M_{Edv} = 3,59 \text{ kNm}$  - accepted reinforcement is sufficient to move the load from the soil.

### 3.7.5 Reinforcements for foundations

The first step is to select the node under the column from the lowest level and Provided Reinforcement of RC Elements.

In new open there is a possibility to change the dimensions of the foundation, based on architectural design are 90x90x30 cm, thickness of underlay 10 cm.

Based on provisional calculations the dimensions of the foundation are insufficient and show on the Figure 56.

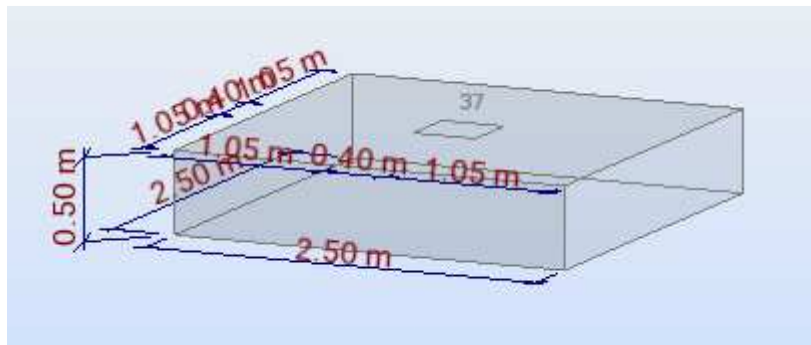


Figure 56 Foundation dimensions

In story parameters was selected the environment class XF2 and structure class S4.

In reinforcement pattern, in Pier ribbon the type is change for third one. In dowel bars was selected Yes for connection to column and stirrups. All options are saved to accelerate design of the other elements. In soils options was selected clayey gravel.

After selecting the options it is possible to make calculation. Selected reinforcement shown in Figure 57.

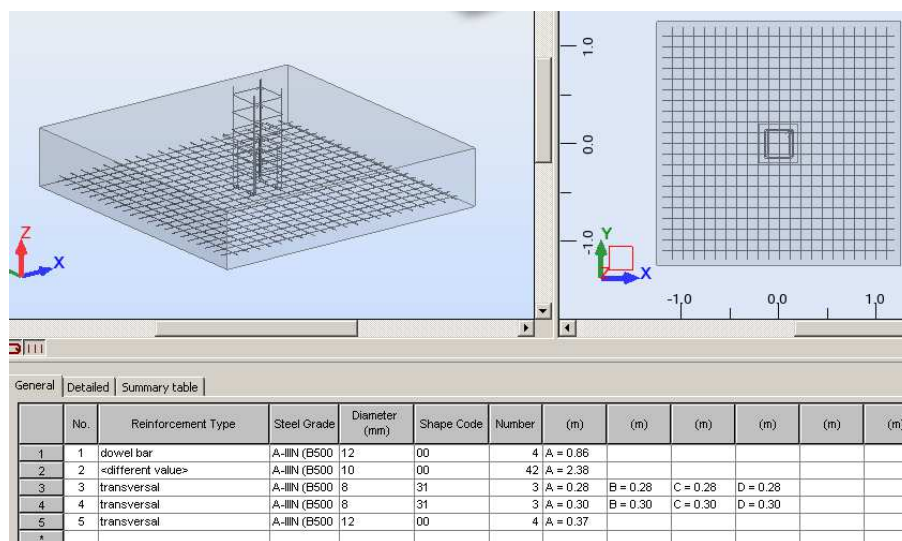
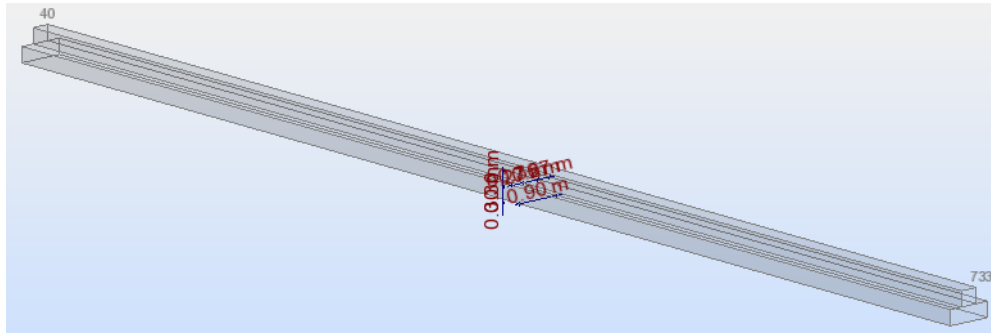


Figure 57 Calculated reinforcement



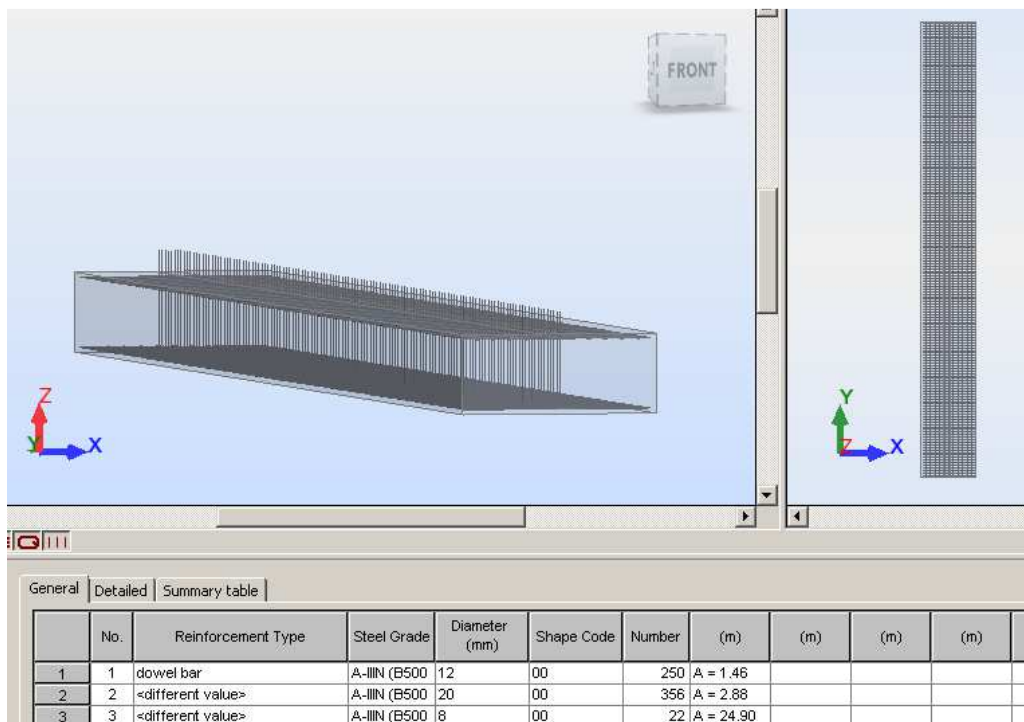
These same steps must be performed for other foundations.

Under the walls were designed continuous footing dimension 0,90x0,50 m as shown Figure 58.



**Figure 58 Continuous footing**

After performing the above steps were obtained required reinforcement as show Figure 59. The basic dimensions were changed by program to 3,00x1,20 m.



**Figure 59 Reinforcement for continuous footing**

## 4 Drawings and update the building model in Revit

The next stage of the work is to make drawings for the designed building. The results obtained in the Robot and calculated reinforcement must be transferred to the Revit program. In the case of version Revit 2015 it is possible to create the reinforcement after installing Revit Extensions 2015. This is add to the program which greatly speeds up the work, the reinforcement can be done for the whole elements in one option window. There is no need to draw every single bar as in the case of AutoCAD. To start drawings should open the program and load family for reinforcement. Then the results from Robot should be send to Revit. After that the model is update, the size of the beam which was change in Robot, now is also change in Revit.

Sending the model and the results to Revit, also the reinforcement should be send, however, it has not been transferred. Based on the information from the Autodesk website it is only possible to transfer the reinforcement for columns and beams, not for all elements. Also during sending the building model the openings in slab are omitted. In the Robot program for each element was calculated required reinforcement. In case of implementation of the reinforcement elements of the same group will be accepted the same, this will allows for a easier connection of elements for the different floors and avoid errors that may appear on the construction site. After updating the model in Revit can be noticed some errors in displaying of certain elements. Load that were created in the Robot also been transferred to the Revit program.

### 4.1 Drawings for beams

- Select the beam in the building model

Based on the calculations from Robot, the required reinforcement is for main 8Ø12 and for stirrups 3Ø8.

In structure ribbon select the reinforcement and Rebar Cover Settings. There was created the new one 40 mm thickness. Thickness was selected due to the fire conditions.

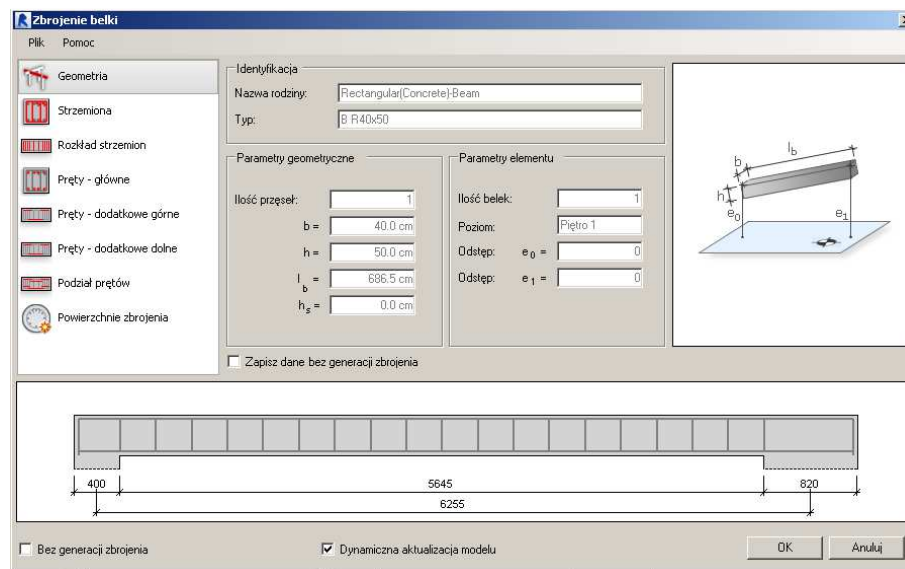
In 3D view was selected the beam and extension ribbon. Next the reinforcement options and beam.

Then will open a new window as show Figure 60, it can be seen that the window of settings in Robot and Revit looks the same, allowing the user to easier learn the program. In first ribbon is the geometry of the element where is not possible to change anything.

In next one for the stirrups was selected bar 8 A-IIIN and the cover from the list.

In the next tab of takes into account compaction the stirrups around the supports and the length of the section  $l = 500$  mm.

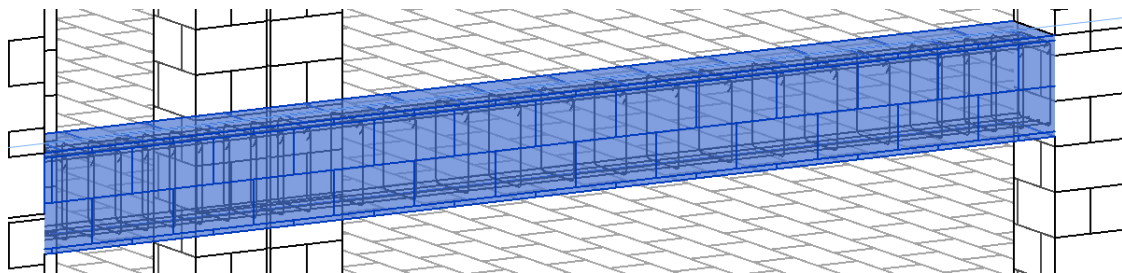
For the main bars was selected 12 A-IIIN, for top reinforcement 4 elements and for bottom also 4.



**Figure 60 Reinforcement settings**

After selecting all options the file would be save to copy the same properties to another elements and press ok to generate the reinforcement. In case of selecting the previously saved parameters of reinforcement for another elements with different geometry, reinforcement accordingly adapts. Also in case of any change, the reinforcement will be updated automatically, that allows to avoid unnecessary errors.

For the rest beams would be selected the previous options.



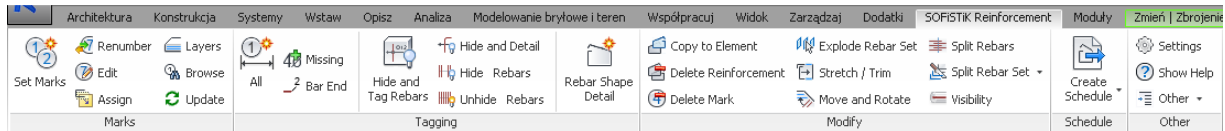
**Figure 61 Reinforcement for beam**

After the creation of reinforcement, could be created a separate section and assigned to them a reinforcement to create documentation.

To create technical drawings, it must be selected the appropriate sections for the beam. Selected scale for sections 1:20. After creating a new sheet the two views of element will be added.

For faster execution of drawings was installed SOFiSTiK Reinforcement Detailing 2015, it is app for Revit which allows as show Figure 62 for rebar shape detailing or create schedule of

reinforcement based on the 3D building model. The free licence is for 30 days and it is available on Autodesk Exchange Apps.



**Figure 62 SOFiSTiK app**

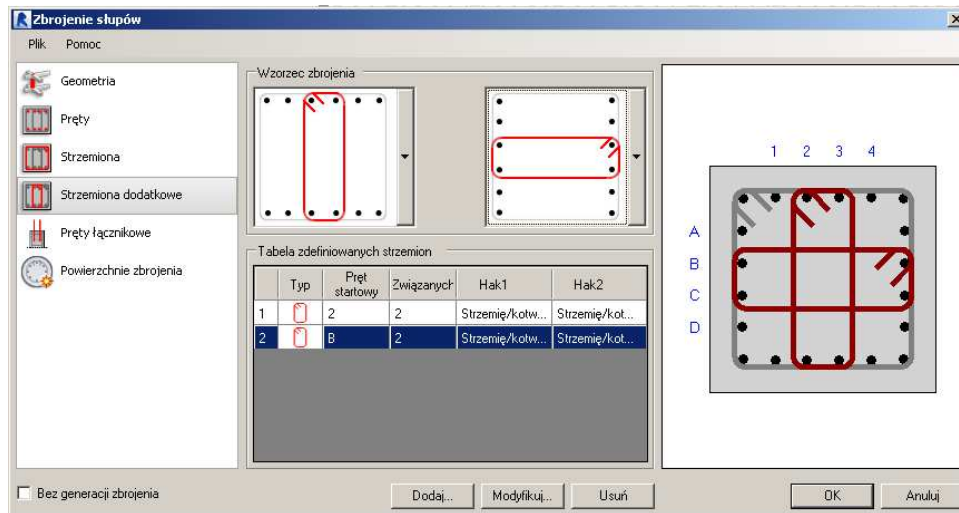
After selecting one of the views using Set Marks and Assign options, they will be numbered bars with the same shape. Then by choosing All Tagging will be added description including number, amount of bars and diameter. In the case where not all the bars have been described it is possible to press Missing Tagging. The next step is Rebar Shape Detail which allows to select the bars and copy the shape and the description. The same tasks must be performed for the second view, which is a cross section through the beam. The final step is to make the schedule of steel. After selecting Create Schedule from ribbon the schedule will open on new window. Then the schedule would be insert to the sheet. In the app settings can be chosen whether the schedule would be generated for one sheet or for the whole building, in this case was selected the first option in order to be able to check the results. In the drawing is required to have the dimensioning of the beam and adding descriptions. It may notice the errors in the SOFiSTiK app, amount of reinforcement in the schedule does not match with the actual number, the amount of the main reinforcement was 4 bars for top and 4 bars for bottom reinforcement, on the schedule the number is 5 for top and 2 for bottom. This may cause a mistake in the process of creating a cost estimate and later ordering reinforcement on construction site.

The last step is to print the drawing from the Revit program.

## 4.2 Drawings for columns

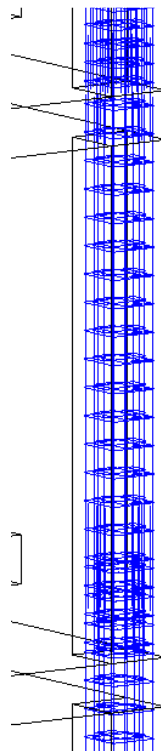
Another element for which would be made the drawings is column. The steps are similar to those of the beam. In extensions ribbon for reinforcement would be selected column. Based on the result and selected reinforcement from the Robot program would be created drawing. The tab geometry could not be changed, the program reads all dimensions, for which would be generated reinforcement.

For column was selected the main bar 12Ø16 and stirrups Ø8 with extra stirrups as show Figure 63.



**Figure 63** Stirrups for column

For the lower floor should take into account the connection of columns as show Figure 64.



**Figure 64** Reinforcement for column

After selecting all of the options will be saved in order to assign them to other columns taking into account the column has to be connected or no. This approach results in a higher cost of the material, however, it will avoid errors and speed up work on construction sites.

In case of selecting a greater number of elements that will have the same reinforcement, appears the message that the elements have different dimensions and it is not possible to generate reinforcement. Even though the columns have the same size, which considerable

slows down the work. Another mistakes occurs during the generation, is that the reinforcement for a few columns were created outside of the element. For this elements the reinforcement would be copied from the next column of the same dimensions.

To create the drawing documentation for column the steps are the same like for beam. They will be created corresponding sections, schedule and described the bars for the sheet.

On the drawing would be presented the column from the foundation to the roof.

### 4.3 Drawings for slabs

The whole slab will be divided into areas which will be the same reinforcement, its value will be selected on the basis of reinforcement maps as show Figure 46.

The Figure 65 shows the distribution for the slab above the last storey.

Separately for each field is selected reinforcement, bar spacing and their diameter. The cover is 40 mm.

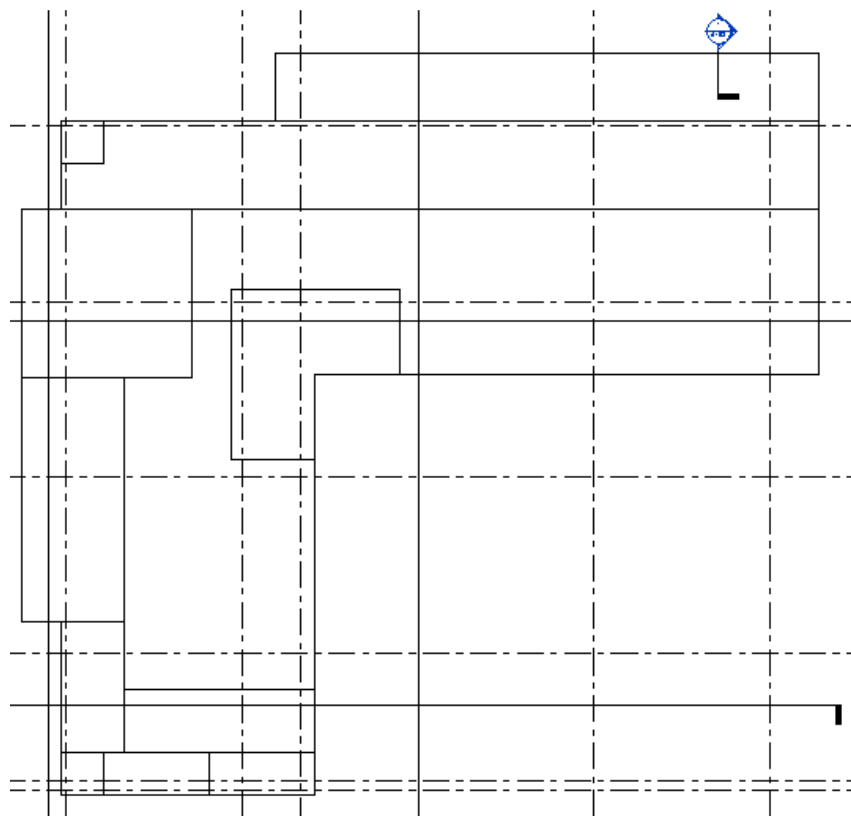
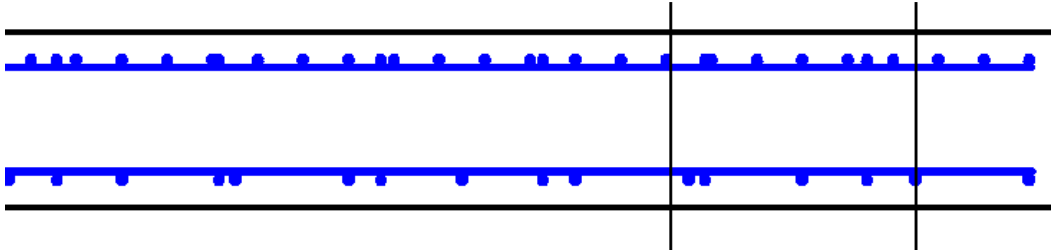


Figure 65 Division slab

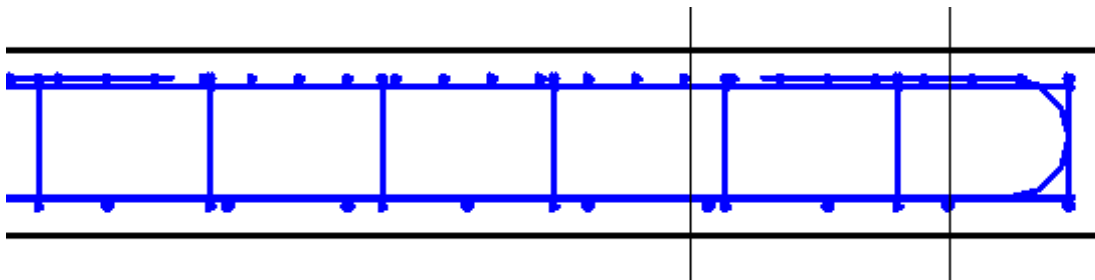
In the case of surface reinforcement bars are straight as show Figure 66, on the edges the bottom bars should be bent. The length of bend should be two thick of slab,  $2 \cdot h = 2 \cdot 30 =$

60cm. In the Revit program is not possible to change the shape of bars for surface reinforcement. The reinforcement of single bars in case of slab is not possible.



**Figure 66 Reinforcement on the edge**

One of the solution to this problem is to add hooks for bottom bars. To do this, select the area of reinforcement, in properties window for the bottom bars change the type of main hook from lack for standard - 180 degrees. Then select the modified bar and select edit type, for a standard diameter of bend hook was changed the value to 180 mm. For length was pressed edit and change the value for 600 mm. After changes reinforcement looks like Figure 67.



**Figure 67 Correct bars on the edge**

These changes should be applied to all surface of reinforcement for the slab. The downside of this method is that changes are implement in both ends of the bar, there is no possibility of introducing a bent only on one side.

To perform the documentation drawings, the first step is to create a new sheets, one for top reinforcement and one for bottom reinforcement.

They will be created two separate plans to show the bottom and top reinforcement.

Because of the better visibility in the drawing for each area of the reinforcement, will be displayed only the middle bar.

In case of reinforcement of slabs for the automatic marking the bars, like for columns, causes an error, the program do not see all the elements, and appears the message that the elements are situated on another sheet.

To describe reinforcement would be selected option Hide and Tag Rebars. Not all bars are numbered, but it is possible to manually perform this. In the specification is the number of bars and their diameter as show Figure 68. Scale of plans was selected 1:200. On the sheet is

also section through the slab only with top or bottom reinforcement with scale 1:100, for which the bars would be described by the option All tagging.

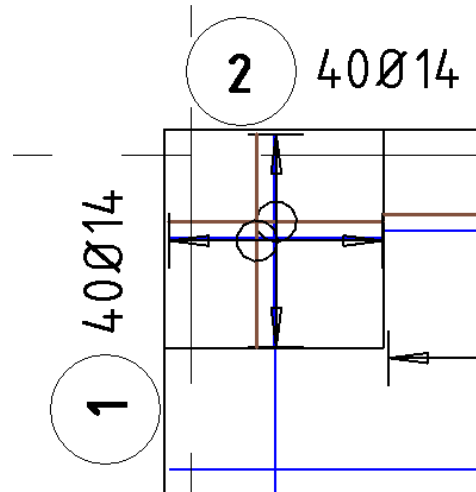


Figure 68 Description of bars

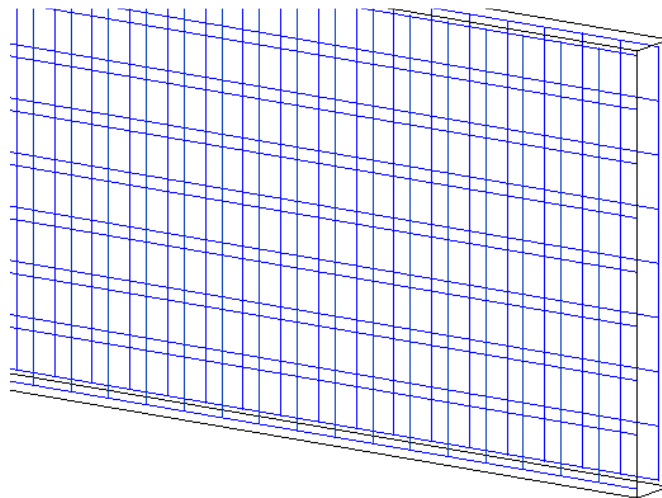
Then the next step is to generate and insert the schedule. At the end must be added the missing descriptions and dimensions.

#### 4.4 Drawings for walls

As for reinforcement for slab would be created for wall. First step is to select the wall and at ribbon which will show select surface. For better work the other elements can be hidden, it will allow for more accurate select the reinforcement area. Before for the element was selected the cover 40 mm. To determine the area of a rectangle is marked along the edge of the wall.

Then reinforcement spacing will be chosen on the basis of calculations made in the previous chapter and bars 12 A-IIIN. (present the Figure 69) The same steps must be repeated for the remaining walls and then perform in Revit drawings documentation. Scale of plans was selected 1:100 and for cross section 1:20.





**Figure 69 Reinforcement for walls**

Also as for the reinforcement of slab all bars except the middle would be hidden. Using tools Rebar Shape Detail bars will be pull out and described, in the case of describing the bars in the wall the program does not see reinforcement in one direction. Then they would be described the bars in the cross section through the wall. As for previous drawing, generate the schedule and insert them into the sheet, dimension the wall and add descriptions.

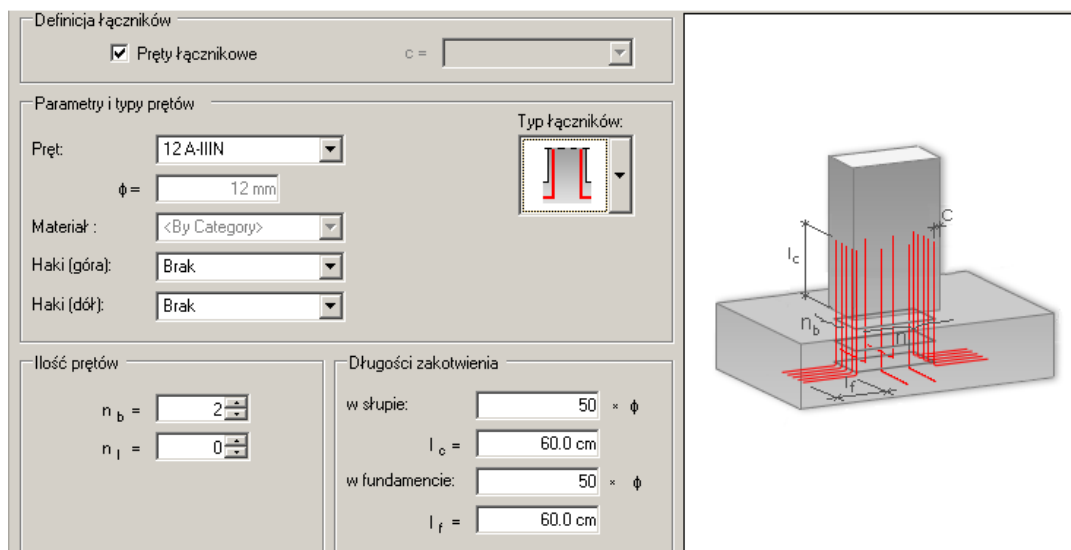
#### **4.5 Drawings for foundations**

The last structural element in the Building model are the foundations. The model includes foundation under the column and continuous footing under the wall.

Based on the original building model from architect size of foundation was 80x80x30 cm and of continuous footing 90x30 cm, after performing calculations dimensions have changed for 200x200x50 cm for column outside the building, 250x250x100 cm for the remaining columns and 350x100 cm.

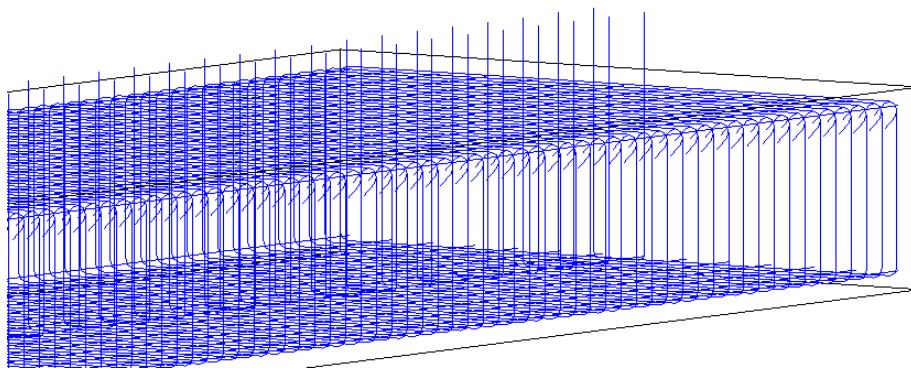
In the Structure ribbon has been added the new thickness of the cover for the foundation, thick 60 mm .

After selecting one of the foundation from the extensions ribbon would be selected required element to generate a reinforcement. In the window with options based on calculations made in the Robot program for bottom bars was selected the right cover previously saved, bars 12 A-IIIN and bar spacing  $s = 150$  mm in both direction. For the top bars the reinforcement is the same as bottom. It is also necessary to take into account for the element the possibility of connection to the column using dowels as show Figure 70 and the stirrups 8 A-IIIN.



**Figure 70 Dowels for foundation**

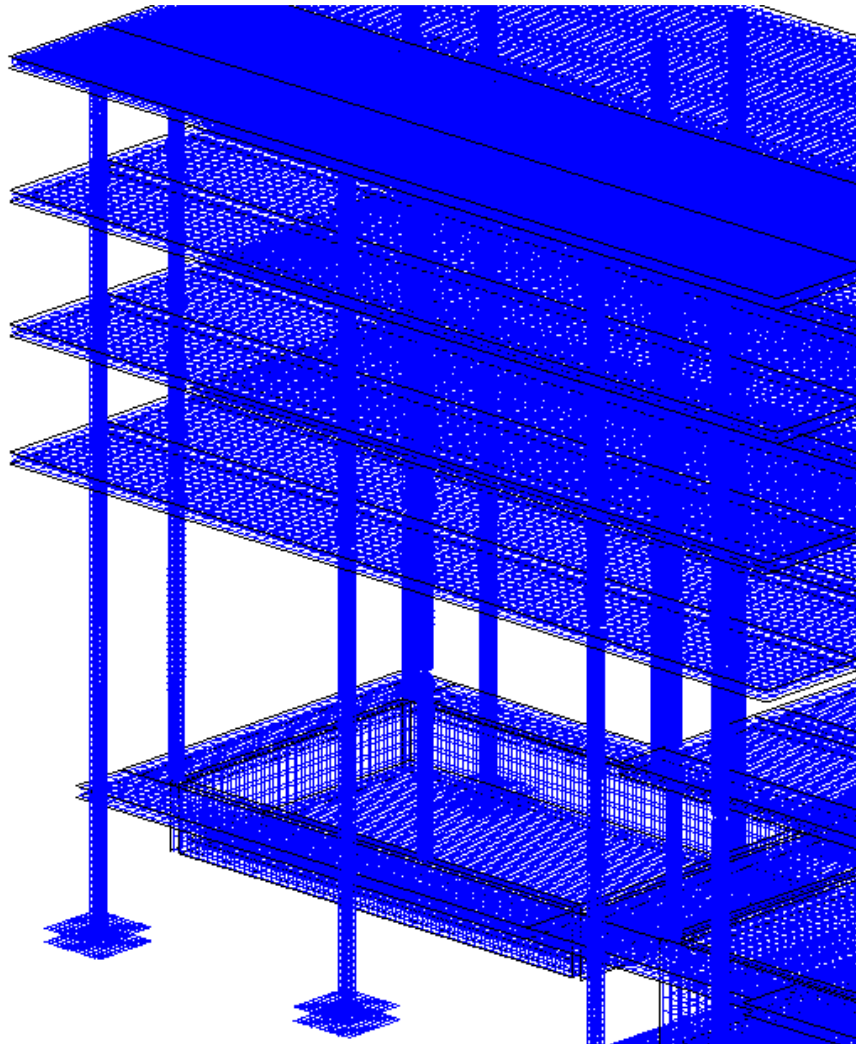
In case of continuous footing for main bars was selected 18 A-IIIIN and  $s_s = 130$  mm. Longitudinal bars 8 A-IIIIN with bottom number  $n_b = 14$  and top  $n_T = 14$ . For connection to the wall applied dowels 12 A-IIIIN with spacing  $s = 300$  mm and length  $l_c = 30$  cm. Figure 71 shows the generated reinforcement after setting all options.



**Figure 71 Reinforcement for continuous footing**

Drawing documentation will be made for one of the foundation. In order to perform it, should be done as in the case of beam.

After perform reinforcement for all structural elements in the building, the model looks like Figure 72.



**Figure 72 Reinforcement of the building**

## 5 Conclusion

The BIM technology is a new and rapidly disseminating method in the design process. At the stage of preliminary design the advantages is the ability to draw ready-made elements that have assigned properties, it is not necessary to draw every single line like in AutoCAD. Also there is no need to make changes to all of drawings, the model is up date all the time which greatly speed up the work and allows to avoid errors. The big advantage is that a single model contains all the necessary information.

The ability to create spatial model allows to view the building before it is executed, it is possible to consider various options for the building. It is also allows to show the model to people who have no knowledge of design. Using the link in Revit the communication between architect and constructor is better.

Possibility of sending the entire structural model for the Robot program allows for much faster operation, it was not necessary to create a model from scratch. The program allows to generate the simulation from wind loads and generate load from the ground.

During the execution drawing documentation, there is no need to draw all the reinforcing bars separately, based on the categories into which include the element it is possible to generate reinforcement and drawings, in order to perform the descriptions and the schedule it was necessary to install additional applications. The disadvantage of SOFiSTiK app is that there is no student version which would be available for free. Installing that app also significantly slowed down the work of Revit program. In case of performing descriptions no all of them were detected by the program, some of them have wrong description and in the schedule is the wrong number of elements. It is important to keep control of the results obtained, it can not just rely on software.

The next problem was to send a model from the ArchiCAD to Revit. The process of creating the IFC file in case of big model takes a lot of time. After opening the model in Revit, the individual elements were not assigned to any of the properties, the reason may be the lack of the library from ArchiCAD. In order to fix this error and accelerate the work was created the link between the original model and a new one where was copy only the structural model.

The process of sending the model to the Robot program for this version was only possible thanks to use intermediate file. In the model in Revit connection between the elements is pinned, which should be change in the Robot program.

Also was noticed error in the analytical model, In the Revit program all the elements were connected to each other, after sending the model in Robot was a lot of mistakes. After the corrected the building model error message appears - discontinuities mesh, the walls, even after manually entering the coordinates ends, the elements do not connected to each other at the same point causing errors. The drawback of the program was the lack of opportunities to generate reinforcement for walls and for slab was not possible to change some settings.

After updating the model in Revit reinforcement was not transferred. During the again try of updating the model appears the message about error. While sending the model to a new file in Revit the holes were omitted, this step has been made in order to check whether it is possible to send reinforcement.

An important aspect which may discourage the introduction of BIM technology is the high price of software, but it is a fast growing technology and the errors that appear in the programs are corrected in the next versions.

## 6 Attachment

Notes calculated from the Robot program .

### 6.1 Robot note for Beam

<b>1</b>	<b>Level:</b>			
	• Name	:	Pietro 3	
	• Reference level	:	---	
	• Maximum cracking	:	0.40 (mm)	
	• Exposure	:	XC1	
	• Concrete creep coefficient	:	$\phi_{\pi}$ = No results	
	• cement class	:	N	
	• Concrete age (loading moment)	:	28 (days)	
	• Concrete age	:	50 (years)	
	• Structure class	:	S4	
	• Fire resistance class	:	R 120(PN-EN 1992-1-2)	
<b>2</b>	<b>Beam: Beam223</b>			<b>Number:</b>
	<b>1</b>			
	<b>2.1</b>	<b>Material properties:</b>		
	• Concrete	:	C45/55	$f_{ck} =$
	45.00 (MPa)			
			Rectangular stress	
	distribution [3.1.7(3)]			
	Density	:	2501.36 (kg/m <sup>3</sup> )	
	Aggregate size	:	20.0 (mm)	
	• Longitudinal reinforcement:	:	A-IIIN (B500SP)	$f_{yk} =$
	500.00 (MPa)			
			Horizontal branch of the	
	stress-strain diagram			
			Ductility class : C	
	• Transversal reinforcement:	:	A-IIIN (B500SP)	$f_{yk} =$
	500.00 (MPa)			
			Horizontal branch of the	
	stress-strain diagram			
			Ductility class : C	
	• Additional reinforcement:	:	A-IIIN (B500SP)	$f_{yk} =$
	500.00 (MPa)			
			Horizontal branch of the	
	stress-strain diagram			
	<b>2.2</b>	<b>Geometry:</b>		
	2.2.1	Span	Position	L.supp.
				L
				R.supp.
				(m)
				(m)
				(m)
		<b>P1</b>	<b>Span 0.40</b>	<b>5.65</b>
				<b>0.82</b>
			Span length: $L_o = 6.26$ (m)	
			Section from 0.00 to 5.65 (m)	

40.0 x 50.0 (cm)  
without left slab  
without right slab

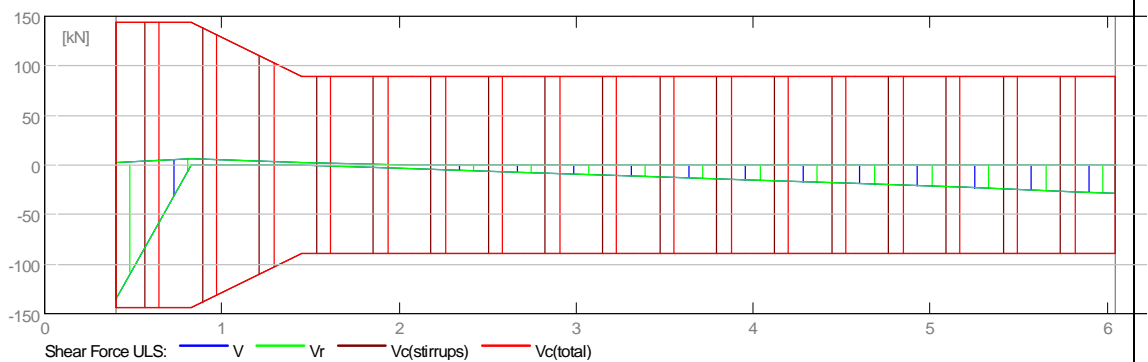
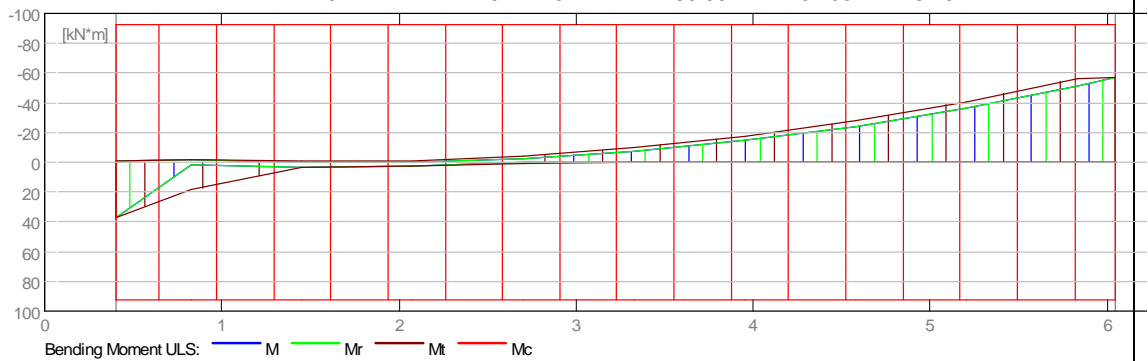
**2.3 Calculation options:**

- Regulation of combinations : PN-EN 1990:2004
- Calculations according to : PN-EN 1992-1-1:2008
- Seismic dispositions : No requirements
- Precast beam : no
- Cover : bottom c = 4.6 (cm)  
: side c1= 5.6 (cm)  
: top c2= 4.6 (cm)
- Cover deviations : Cdev = 1.0(cm), Cdur = 0.0(cm)
- Coefficient  $\beta_2 = 0.50$  : long-term or cyclic load
- Method of shear calculations : strut inclination

**2.4 Calculation results:**

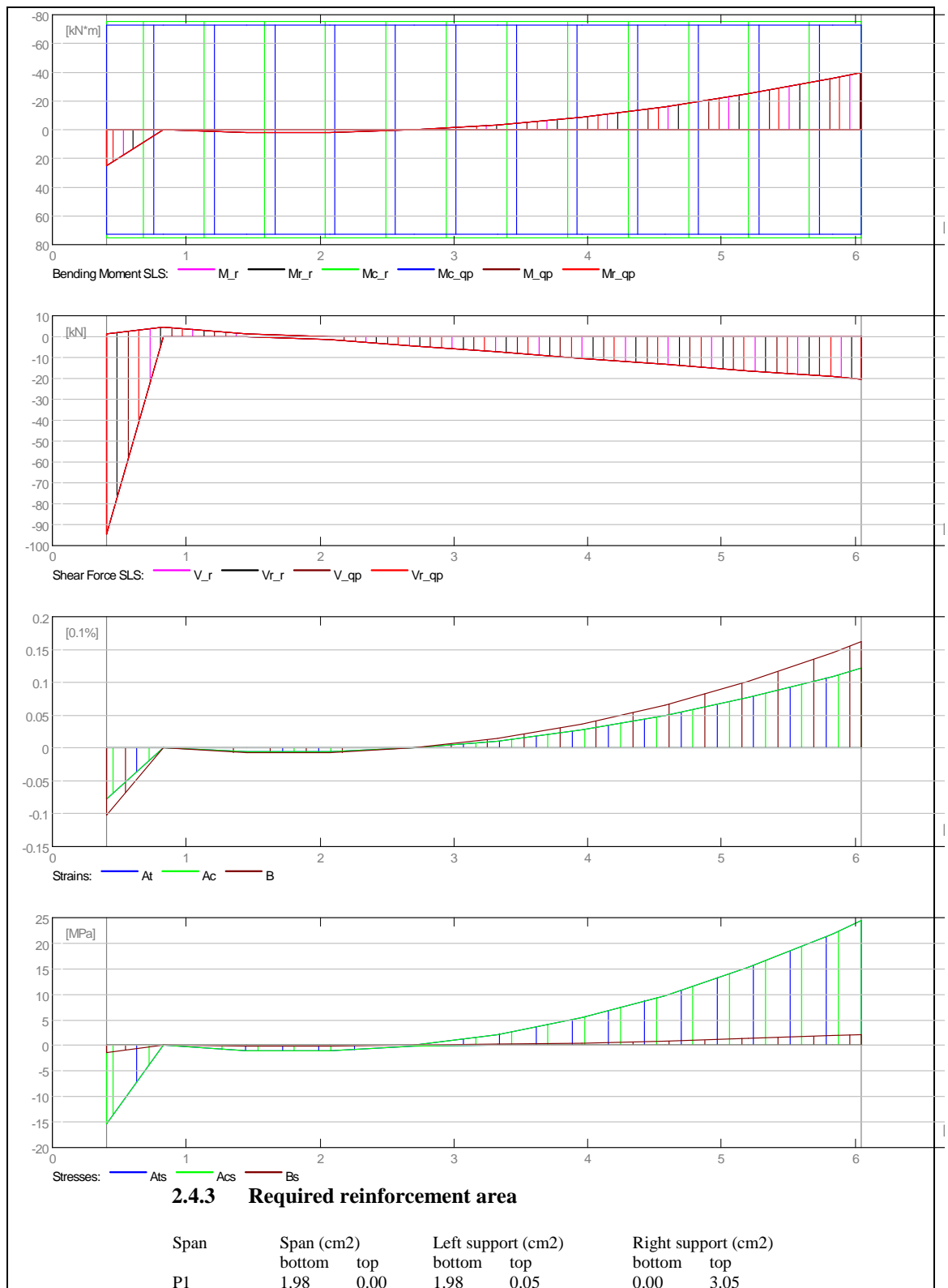
**2.4.1 Internal forces in ULS**

Span	Mt max. (kN*m)	Mt min. (kN*m)	MI (kN*m)	Mr (kN*m)	Ql (kN)	Qr (kN)
P1	37.14	-27.76	37.14	-56.96	-134.96	-28.16

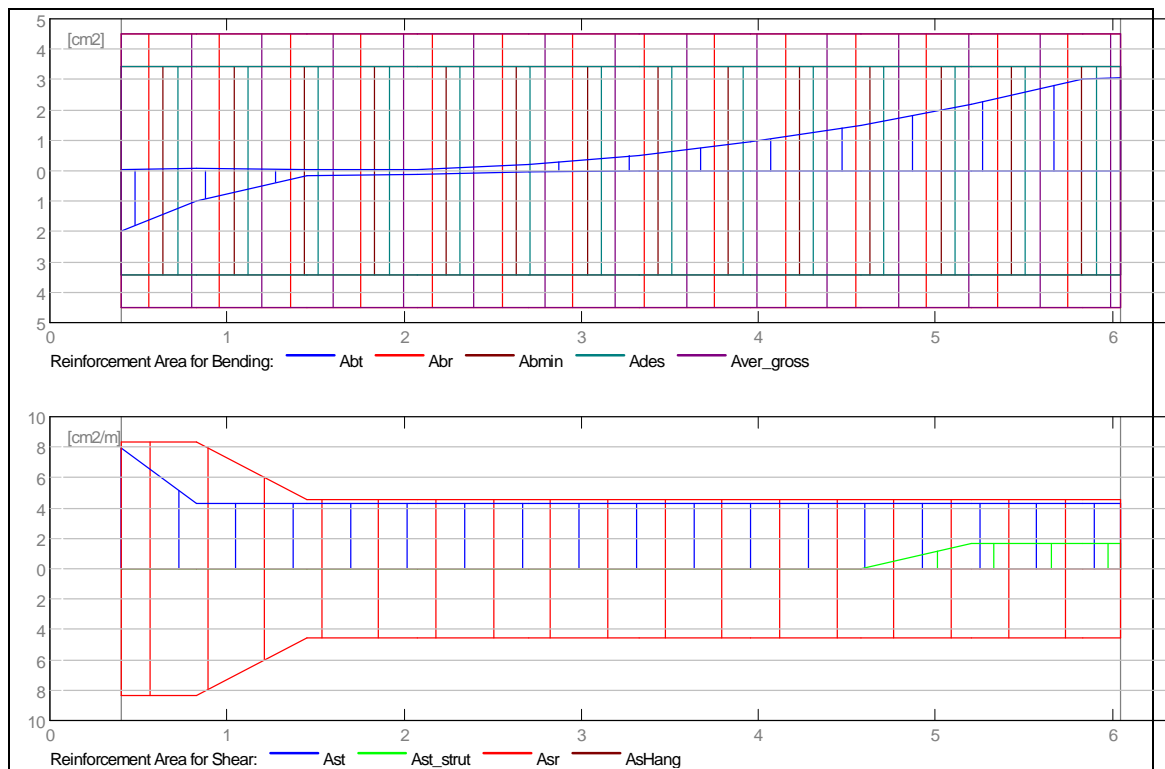


**2.4.2 Internal forces in SLS**

Span	Mt max. (kN*m)	Mt min. (kN*m)	MI (kN*m)	Mr (kN*m)	Ql (kN)	Qr (kN)
P1	25.14	-15.99	25.14	-39.92	-94.66	-20.23







**2.4.4 Fire resistance**

Fire resistance :R 120(PN-EN 1992-1-2)  
 Calculations according to :PN-EN 1992-1-2  
 Estimation in accordance with section 5. Tabulated data.  
 Number of sides exposed to fire :3  
 Web type :WA  
 Beam type :freely supported  
 b\_min = 0.20(m)  
 a\_min = 0.05(m)

**2.4.5 Deflection and cracking**

wt(QP) Total due to quasi-permanent combination  
 wt(QP)dop Allowable due to quasi-permanent combination  
 Dwt(QP) Deflection increment from the quasi-permanent load combination after erecting a structure.  
 Dwt(QP)dop Admissible deflection increment from the quasi-permanent load combination after erecting a structure.  
 wk - width of perpendicular cracks

Span	wt(QP) (cm)	wt(QP)dop (cm)	Dwt(QP) (cm)	Dwt(QP)dop (cm)	wk (mm)
P1	0.0	2.5	0.0	1.3	0.0

**2.5 Theoretical results - detailed results:**

**2.5.1 P1 : Span from 0.40 to 6.05 (m)**

Abscissa (m)	ULS		SLS		A bottom (cm2)	A top (cm2)
	M max. (kN*m)	M min. (kN*m)	M max. (kN*m)	M min. (kN*m)		
0.40	37.14	-0.99	25.14	-0.01	1.98	0.05
0.83	18.52	-1.53	0.00	-0.04	0.98	0.08
1.45	2.97	-0.50	1.77	0.00	0.15	0.03
2.08	2.60	-0.74	1.80	0.00	0.13	0.04

2.70	0.79	-3.93	0.04	0.00	0.04	0.20
3.33	0.00	-9.66	0.00	-3.52	0.00	0.51
3.95	0.00	-17.51	0.00	-8.86	0.00	0.93
4.58	0.00	-27.76	0.00	-15.99	0.00	1.48
5.20	0.00	-40.85	0.00	-24.90	0.00	2.18
5.83	0.00	-56.60	0.00	-35.61	0.00	3.04
6.05	0.00	-56.96	0.00	-39.92	0.00	3.05

Abscissa (m)	ULS	SLS	afp (mm)
	V max. (kN)	V max. (kN)	
0.40	-134.96	-94.66	0.0
0.83	6.12	4.48	0.0
1.45	2.09	1.52	0.0
2.08	-3.45	-1.44	0.0
2.70	-6.87	-4.40	0.0
3.33	-10.73	-7.36	0.0
3.95	-14.74	-10.33	0.0
4.58	-18.75	-13.29	0.0
5.20	-22.76	-16.25	0.0
5.83	-26.78	-19.21	0.0
6.05	-28.16	-20.23	0.0

**2.6 Reinforcement:**

**2.6.1 P1 : Span from 0.40 to 6.05 (m)**

**Longitudinal reinforcement:**

- bottom (A-IIIN (B500SP))  
4 φ12 l = 6.25 from 0.10 to 6.35
- support (A-IIIN (B500SP))  
4 φ12 l = 6.43 from 0.05 to 6.47

**Transversal reinforcement:**

- main (A-IIIN (B500SP))  
stirrups 27 φ8 l = 1.49  
e = 1\*0.11 + 4\*0.12 + 22\*0.22 (m)
- pins 27 φ8 l = 1.49  
e = 1\*0.11 + 4\*0.12 + 22\*0.22 (m)

**3 Material survey:**

- Concrete volume = 1.37 (m3)
- Formwork = 9.52 (m2)
- Steel A-IIIN (B500SP)
  - Total weight = 60.89 (kG)
  - Density = 44.35 (kG/m3)
  - Average diameter = 10.2 (mm)
  - Survey according to diameters:

Diameter (mm)	Length (m)	Weight (kG)	Number (No.)	Total weight (kG)
8	1.49	0.59	27	15.86
12	6.25	5.55	4	22.19
12	6.43	5.71	4	22.84

## 6.2 Robot note for Column

<b>1</b>	<b>Level:</b>		
	• Name	: Piętro 4	
	• Reference level	: 15.70 (m)	
	• Concrete creep coefficient	: $\Phi_p = 1.77$	
	• cement class	: N	
	• Environment class	: XC1	
	• Structure class	: S4	
<b>2</b>	<b>Column: Column154</b>		<b>Number: 1</b>
	<b>2.1 Material properties:</b>		
	• Concrete	: C45/55	$f_{ck} = 45.00$ (MPa)
	Unit weight	: 2501.36 (kG/m3)	
	Aggregate size	: 20.0 (mm)	
	• Longitudinal reinforcement:	: A-IIIN (B500SP)	$f_{yk} = 500.00$ (MPa)
	Ductility class	: C	
	• Transversal reinforcement:	: A-IIIN (B500SP)	$f_{yk} = 500.00$ (MPa)
	<b>2.2 Geometry:</b>		
	2.2.1 Rectangular	40.0 x 40.0 (cm)	
	2.2.2 Height: L	= 3.45 (m)	
	2.2.3 Slab thickness	= 0.30 (m)	
	2.2.4 Beam height	= 0.30 (m)	
	2.2.5 Cover	= 5.0 (cm)	
	<b>2.3 Calculation options:</b>		
	• Calculations according to	: PN-EN 1992-1-1:2008	
	• Seismic dispositions	: No requirements	
	• Precast column	: no	
	• Pre-design	: no	
	• Slenderness taken into account	: yes	
	• Compression	: with bending	
	• Ties	: to slab	
	• Fire resistance class	: R 120	
	<b>2.4 Calculation results:</b>		
	Safety factors $R_d/E_d = 1.42 > 1.0$		
	<b>2.4.1 Fire resistance</b>		
	Calculations according to: : PN-EN 1992-1-2:2008		
	•	Estimation in accordance with section 5. Tabulated data.	
	•	Number of sides exposed to fire:	: >1
	•	Reduction factor in fire situation	: according to combinations
	•	Calculation method	: A
	•	Effective column length in fire conditions	: $l_{0y,fi} = 3.45$ (m)
	•	Effective column length in fire conditions	: $l_{0z,fi} = 0.00$ (m)
	•	Load level reduction factor	: $\mu_{fi} = 0.00$
	•	Combination for load level reduction factor	: -

- Ratio :  $\omega = 0.038$
- Number of main bars : 4
- $R_a = 1,6 * (a * 1000 - 30) = 59.20$
- $R_l = 9,6 * (5 - l_{o,fi}) = 14.88$
- $R_b = 90 * b' = 36.00$
- $R_n = 0.00$
- $R_{\eta fi} = 83 * (1 - \mu * (1 + \omega)) / (0.85 / acc + \omega) = 83.00$
- $R = 282, R_{282} \geq R_{120}$

#### 2.4.2 ULS/ALS Analysis

Design combination: KOMB5 (B)

Combination type: ULS

Internal forces:

$$N_{sd} = 363.51 \text{ (kN)} \quad M_{sdy} = -80.27 \text{ (kN*m)} \quad M_{sdz} = 25.25 \text{ (kN*m)}$$

Design forces:

Lower node

$$N = 363.51 \text{ (kN)} \quad N^*e_{totz} = -83.40 \text{ (kN*m)} \quad N^*e_{toty} = 25.25 \text{ (kN*m)}$$

Eccentricity:	ez (My/N)	ey (Mz/N)
Static	eEd: -22.1 (cm)	6.9 (cm)
Imperfection	ei: 0.9 (cm)	0.0 (cm)
Initial	e0: -21.2 (cm)	6.9 (cm)
Minimal	emin: 2.0 (cm)	2.0 (cm)
Total	etot: -22.9 (cm)	6.9 (cm)

##### 2.4.2.1. Detailed analysis-Direction Y:

###### 2.4.2.1.1 Slenderness analysis

Non-sway structure

L (m)	Lo (m)	$\lambda$	$\lambda_{lim}$	
3.45	3.45	29.88	152.26	Short column

###### 2.4.2.1.2 Buckling analysis

$$M_2 = 75.47 \text{ (kN*m)} \quad M_1 = -80.27 \text{ (kN*m)}$$

Case: Cross-section at the column end (Lower node), Slenderness not taken into account

$$M_0 = -80.27 \text{ (kN*m)}$$

$$e_a = \theta_1 * l_0 / 2 = 0.9 \text{ (cm)}$$

$$\theta_1 = \theta_0 * \alpha_{\eta} * \alpha_m = 0.01$$

$$\theta_0 = 0.01$$

$$\alpha_h = 1.00$$

$$\alpha_m = (0,5(1+1/m))^{0.5} = 1.00$$

$$m = 1.00$$

$$M_a = N * e_a = 3.14 \text{ (kN*m)}$$

$$M_{Edmin} = 7.27 \text{ (kN*m)}$$

$$M_{0Ed} = \max(M_{Edmin}, M_0 + M_a) = -83.40 \text{ (kN*m)}$$

##### 2.4.2.2. Detailed analysis-Direction Z:

$$M_2 = 25.25 \text{ (kN*m)} \quad M_1 = -24.87 \text{ (kN*m)}$$

Case: Cross-section at the column end (Lower node), Slenderness not taken into account

$$M_0 = 25.25 \text{ (kN*m)}$$

$$e_a = 0.0 \text{ (cm)}$$

$$M_a = N * e_a = 0.00 \text{ (kN*m)}$$

$$M_{Edmin} = 7.27 \text{ (kN*m)}$$

$$M_{0Ed} = \max(M_{Edmin}, M_0 + M_a) = 25.25 \text{ (kN*m)}$$

#### 2.4.3 Reinforcement:

Real (provided) area

$$A_{sr} = 4.52 \text{ (cm}^2\text{)}$$

Ratio:

$$\rho = 0.28 \%$$

#### 2.5 Reinforcement:

<b>Main bars (A-IIIN (B500SP)):</b>				
• 4 $\phi$ 12      l = 3.40 (m)				
<b>Transversal reinforcement: (A-IIIN (B500SP)):</b>				
stirrups:      16 $\phi$ 8      l = 1.30 (m)				
pins            16 $\phi$ 8      l = 1.30 (m)				
<b>3</b>	<b>Material survey:</b>			
	• Concrete volume      = 0.50 (m3)			
	• Formwork      = 5.04 (m2)			
	• Steel A-IIIN (B500SP)			
	• Total weight      = 20.27 (kG)			
	• Density      = 40.21 (kG/m3)			
	• Average diameter      = 9.6 (mm)			
	• Reinforcement survey:			
	Diameter	Length	Weight	Number
		(m)	(kG)	(No.)
	8	1.30	0.51	16
	12	3.40	3.02	4
				Total weight
				(kG)
				8.19
				12.08

**6.3 Robot note for slab**

<b>1. Slab: Plate249 - Panel no. 249</b>	
<b>1.1. Reinforcement:</b>	
• Type	: Strop zelbetowy
• Main reinforcement direction	: 0°
Main reinforcement grade	: A-IIIN (B500SP); Characteristic strength = 500.00 MPa
	Horizontal branch of the stress-strain diagram
• Ductility class	: C
• Bar diameters	bottom d1 = 1.2 (cm) d2 = 1.2 (cm) top d1 = 1.2 (cm) d2 = 1.2 (cm)
• Cover	bottom c1 = 3.0 (cm) top c2 = 3.0 (cm)
• Cover deviations	Cdev = 1.0(cm), Cdur = 0.0(cm)
<b>1.2. Concrete</b>	
Class	: C45/55; Characteristic strength = 45.00 MPa Rectangular stress distribution [3.1.7(3)]
• Density	: 2501.36 (kG/m3)
• Concrete creep coefficient	: 1.67
• cement class	: N
<b>1.3. Hypothesis</b>	
• Calculations according to	: PN-EN 1992-1-1:2008
• Method of reinforcement area calculations	: analytical
• Allowable cracking width	
- upper layer	: 0.40 (mm)

- lower layer : 0.40 (mm)
- Allowable deflection : 3.0 (cm)
- Verification of punching : yes
- Exposure
- upper layer : X0
- lower layer : X0
- Calculation type : simple bending
- Structure class : S4

**1.4. Slab geometry**

Thickness 0.30 (m)

**Contour:**

edge	beginning		end		length (m)
	x1	y1	x2	y2	
1	0.00	22.09	0.00	25.30	3.21
2	0.00	25.30	-2.64	25.30	2.64
3	-2.64	25.30	-2.64	21.19	4.11
4	-2.64	21.19	-4.51	21.19	1.88
5	-4.51	21.19	-4.51	1.68	19.52
6	-4.51	1.68	-2.63	1.68	1.88
7	-2.63	1.68	-2.63	0.00	1.68
8	-2.63	0.00	0.00	0.00	2.63
9	0.00	0.00	0.48	0.00	0.48
10	0.48	0.00	0.48	-1.22	1.22
11	0.48	-1.22	1.79	-1.22	1.31
12	1.79	-1.22	1.79	-4.73	3.51
13	1.79	-4.73	0.53	-4.73	1.26
14	0.53	-4.73	0.53	-6.51	1.78
15	0.53	-6.51	7.94	-6.51	7.41
16	7.94	-6.51	9.32	-6.51	1.38
17	9.32	-6.51	9.32	13.35	19.86
18	9.32	13.35	33.08	13.35	23.77
19	33.08	13.35	33.08	28.55	15.20
20	33.08	28.55	7.46	28.55	25.62
21	7.46	28.55	7.46	25.30	3.25
22	7.46	25.30	4.50	25.30	2.96
23	4.50	25.30	4.50	22.09	3.21
24	4.50	22.09	0.00	22.09	4.50

**Support:**

n°	Name	dimensions (m)	coordinates		edge
			x	y	
117	point	0.40 / 0.40	30.72	25.05	—
117	point	0.40 / 0.40	30.72	25.05	—
226	point	0.40 / 0.40	14.17	14.04	—
226	point	0.40 / 0.40	14.17	14.04	—
228	point	0.40 / 0.40	22.47	14.04	—
228	point	0.40 / 0.40	22.47	14.04	—
230	point	0.40 / 0.40	30.77	14.04	—
230	point	0.40 / 0.40	30.77	14.04	—
232	point	0.40 / 0.40	8.63	8.50	—
232	point	0.40 / 0.40	8.63	8.50	—
234	point	0.40 / 0.40	8.63	14.04	—
234	point	0.40 / 0.40	8.63	14.04	—
236	point	0.40 / 0.40	8.63	0.20	—
236	point	0.40 / 0.40	8.63	0.20	—
238	point(*)	0.40 / 0.40	-2.43	0.20	—

238	point	0.40 / 0.40	-2.43	0.20	—
238	linear	0.20 / 0.40	-2.43	0.10	—
240	point	0.40 / 0.40	-2.43	16.80	—
240	point	0.40 / 0.40	-2.43	16.80	—
242	point	0.40 / 0.40	16.29	14.72	—
244	point	0.40 / 0.40	5.87	0.20	—
244	point	0.40 / 0.40	5.87	0.20	—
246	point	0.40 / 0.40	5.87	8.50	—
246	point	0.40 / 0.40	5.87	8.50	—
248	point	0.40 / 0.40	-2.43	8.50	—
248	point	0.40 / 0.40	-2.43	8.50	—
250	point	0.40 / 0.40	5.87	16.80	—
250	point	0.40 / 0.40	5.87	16.80	—
254	point	0.40 / 0.40	22.47	16.80	—
254	point	0.40 / 0.40	22.47	16.80	—
256	point	0.40 / 0.40	30.72	16.85	—
256	point	0.40 / 0.40	30.72	16.85	—
260	point	0.40 / 0.40	22.47	25.10	—
260	point	0.40 / 0.40	22.47	25.10	—
262	point	0.40 / 0.40	14.23	25.16	—
262	point	0.40 / 0.40	14.23	25.16	—
264	point	0.40 / 0.40	5.87	-5.85	—
264	point	0.40 / 0.40	5.87	-5.85	—
268	point	0.40 / 0.40	8.63	-5.85	—
268	point	0.40 / 0.40	8.63	-5.85	—
270	point	0.40 / 0.40	22.47	27.86	—
270	point	0.40 / 0.40	22.47	27.86	—
272	point	0.40 / 0.40	10.02	27.86	—
272	point	0.40 / 0.40	10.02	27.86	—
274	point	0.40 / 0.40	30.77	27.86	—
274	point	0.40 / 0.40	30.77	27.86	—
276	point	0.40 / 0.40	14.17	27.86	—
276	point	0.40 / 0.40	14.17	27.86	—
308	point	0.40 / 0.40	5.87	25.10	—
308	point	0.40 / 0.40	5.87	25.10	—
331	point(*)	0.40 / 0.40	-2.43	25.10	—
331	point	0.40 / 0.40	-2.43	25.10	—
350	point	0.40 / 0.40	8.63	-6.27	—
350	point	0.40 / 0.40	8.63	-6.27	—
352	point	0.40 / 0.40	5.87	-6.27	—
352	point	0.40 / 0.40	5.87	-6.27	—
1307	point	0.40 / 0.40	14.23	16.80	—
1307	point	0.40 / 0.40	14.23	16.80	—
7184	linear	0.50 / 0.50	33.08	20.95	—
7103	linear	0.50 / 0.50	11.73	28.55	—
7061	linear	0.50 / 0.50	28.81	28.55	—
7174	linear	0.50 / 0.50	20.27	28.55	—
* - head present					
<b>Column head:</b>					
	<b>support no.</b>	<b>dimensions (m)</b>			
		<b>bottom</b>	<b>top</b>	<b>height</b>	
	238	0.40 / 0.40	1.00 / 1.00	0.50	
	331	0.40 / 0.40	1.00 / 1.00	0.50	
<b>1.5. Calculation results:</b>					

**1.5.1. Maximum moments + reinforcement for bending**

	<b>Ax(+) Ay(-)</b>	<b>Ax(-)</b>	<b>Ay(+)</b>	
Provided reinforcement (cm2/m):	89.36 33.51	73.11	38.30	
Modified required reinforcement (cm2/m):	78.06 30.33	66.26	31.25	
Original required reinforcement (cm2/m):	78.06 30.33	66.26	31.25	
Coordinates (m):	-2.18;0.00	-2.63;24.79	-2.14;25.05	-
	2.63;24.79			

**1.5.2. Maximum moments + reinforcement for bending**

	<b>Ax(+) Ay(-)</b>	<b>Ax(-)</b>	<b>Ay(+)</b>	
Symbol: required area/provided area				
Ax(+) (cm2/m)	<b>78.06/89.36</b> 30.24/89.36	30.24/89.36	28.53/89.36	
Ax(-) (cm2/m)	47.65/73.11 <b>66.26/73.11</b>	<b>66.26/73.11</b>	5.02/73.11	
Ay(+) (cm2/m)	16.46/38.30 9.17/38.30	9.17/38.30	<b>31.25/38.30</b>	
Ay(-) (cm2/m)	4.78/33.51 <b>30.33/33.51</b>	30.33/33.51	4.78/33.51	
	<b>SLS</b>			
Mxx (kN*m/m)	347.63 156.78	-156.78	118.07	-
Myy (kN*m/m)	85.69 127.55	127.55	142.49	
Mxy (kN*m/m)	51.88 42.94	42.94	-77.02	
Nxx (kN/m)	260.32 9.85	-9.85	11.90	-
Nyy (kN/m)	63.92 23.87	23.87	22.83	
Nxy (kN/m)	95.90 13.82	13.82	-12.48	
	<b>ULS</b>			
Mxx (kN*m/m)	496.64 225.08	-225.08	170.26	-
Myy (kN*m/m)	121.52 181.46	181.46	202.98	
Mxy (kN*m/m)	73.89 61.01	61.01	-110.54	
Nxx (kN/m)	390.14 13.64	-13.64	16.64	-
Nyy (kN/m)	96.88 33.48	33.48	31.47	
Nxy (kN/m)	144.83 19.17	19.17	-17.36	
Coordinates (m)	-2.18;0.00	-2.63;24.79	-2.14;25.05	-



2.63;24.79

Coordinates\* (m)

7.90;65.51;15.70

7.45;90.30;15.70

7.94;90.56;15.70

7.45;90.30;15.70

\* - Coordinates in the structure global coordinate system

### 1.5.3. Punching

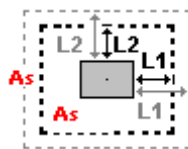
Support no. / Point	Location (m)			Geometry: (m)			h
	x	y		a	b	d	
S2	30.72	25.05	Column		0.40	0.40	-
S3	14.17	14.04	Column		0.40	0.40	-
S4	14.17	14.04	Column		0.40	0.40	-
S5	22.47	14.04	Column		0.40	0.40	-
S6	22.47	14.04	Column		0.40	0.40	-
S7	30.77	14.04	Column		0.40	0.40	-
S8	30.77	14.04	Column		0.40	0.40	-
S10	8.63	8.50	Column		0.40	0.40	-
S11	8.63	14.04	Column		0.40	0.40	-
S12	8.63	14.04	Column		0.40	0.40	-
S14	8.63	0.20	Column		0.40	0.40	-
S17	-2.43	16.80	Column		0.40	0.40	-
S18	16.29	14.72	Column		0.40	0.40	-
S19	5.87	0.20	Column		0.40	0.40	-
S20	5.87	0.20	Column		0.40	0.40	-
S21	5.87	8.50	Column		0.40	0.40	-
S22	5.87	8.50	Column		0.40	0.40	-
S23	-2.43	8.50	Column		0.40	0.40	-
S24	-2.43	8.50	Column		0.40	0.40	-
S26	5.87	16.80	Column		0.40	0.40	-
S28	22.47	16.80	Column		0.40	0.40	-
S30	30.72	16.85	Column		0.40	0.40	-
S32	22.47	25.10	Column		0.40	0.40	-
S34	14.23	25.16	Column		0.40	0.40	-
S35	5.87	-5.85	Column		0.40	0.40	-
S36	5.87	-5.85	Column		0.40	0.40	-
S37	8.63	-5.85	Column		0.40	0.40	-
S38	8.63	-5.85	Column		0.40	0.40	-
S39	22.47	27.86	Column		0.40	0.40	-
S40	22.47	27.86	Column		0.40	0.40	-
S41	10.02	27.86	Column		0.40	0.40	-
S42	10.02	27.86	Column		0.40	0.40	-
S43	30.77	27.86	Column		0.40	0.40	-
S44	30.77	27.86	Column		0.40	0.40	-
S45	14.17	27.86	Column		0.40	0.40	-
S46	14.17	27.86	Column		0.40	0.40	-
S47	5.87	25.10	Column		0.40	0.40	-
S48	5.87	25.10	Column		0.40	0.40	-
S49 Head	-2.43	25.10	Column		0.40/1.00	0.40/1.00	-/-
S50	-2.43	25.10	Column		0.40	0.40	-
S51	8.63	-6.27	Column		0.40	0.40	-
S53	5.87	-6.27	Column		0.40	0.40	-
S54	5.87	-6.27	Column		0.40	0.40	-
S56	14.23	16.80	Column		0.40	0.40	-
P1	33.08	20.95	Force	0.50	0.50	-	-
P2	11.73	28.55	Force	0.50	0.50	-	-
P3	28.81	28.55	Force	0.50	0.50	-	-
P4	20.27	28.55	Force	0.50	0.50	-	-

Support no. / Point	Loads: (kN)		Critical perimeter (m)	Qadm / Q
	Q	Qadm		
S2	809.84	998.00	4.84	1.23 > 1
S3	0.11	594.64	2.42	5177.65 > 1
S4	590.18	590.18	2.42	1.00 > 1
S5	0.09	594.64	2.42	6929.80 > 1
S6	612.32	612.32	2.42	1.00 > 1
S7	0.05	594.64	2.42	11182.09 > 1
S8	491.29	499.00	2.42	1.02 > 1
S10	591.04	591.04	2.42	1.00 > 1
S11	0.07	1189.28	4.84	18042.28 > 1

S12	648.03	998.00	4.84	1.54 > 1
S14	566.95	566.95	2.42	1.00 > 1
S17	1200.62	1200.62	4.84	1.00 > 1
S18	0.05	1189.28	4.84	21915.07 > 1
S19	0.03	1189.28	4.84	42609.53 > 1
S20	872.38	998.00	4.84	1.14 > 1
S21	0.02	1189.28	4.84	64743.64 > 1
S22	1009.11	1009.11	4.84	1.00 > 1
S23	0.03	1189.28	4.84	43914.46 > 1
S24	1227.94	1227.94	4.84	1.00 > 1
S26	915.25	998.00	4.84	1.09 > 1
S28	1096.87	1096.87	4.84	1.00 > 1
S30	914.56	998.00	4.84	1.09 > 1
S32	1149.72	1149.72	4.84	1.00 > 1
S34	1056.71	1056.71	4.84	1.00 > 1
S35	0.03	594.64	2.42	17830.24 > 1
S36	593.44	628.70	2.42	1.06 > 1
S37	90.20	297.32	1.21	3.30 > 1
S38	118.22	249.50	1.21	2.11 > 1
S39	0.10	594.64	2.42	6169.48 > 1
S40	508.14	508.14	2.42	1.00 > 1
S41	0.10	594.64	2.42	6165.06 > 1
S42	548.50	548.50	2.42	1.00 > 1
S43	0.09	594.64	2.42	6817.96 > 1
S44	433.75	499.00	2.42	1.15 > 1
S45	0.18	594.64	2.42	3217.18 > 1
S46	577.29	577.29	2.42	1.00 > 1
S47	886.97	886.97	2.42	1.00 > 1
S48	0.10	594.64	2.42	5812.76 > 1
S49	961.76	961.76	2.42	1.00 > 1
S50	0.03	667.46	2.42	22911.03 > 1
S51	113.89	249.50	1.21	2.19 > 1
S53	525.96	628.70	2.42	1.20 > 1
S54	0.02	594.64	2.42	35044.47 > 1
S56	1111.62	1111.62	4.84	1.00 > 1
P1	0.00	582.36	2.37	295613.81 > 1
P2	0.00	244.35	1.19	115859.43 > 1
P3	0.00	488.70	2.37	231718.86 > 1
P4	0.00	244.35	1.19	115859.43 > 1

**Reinforcement parameters:**

- Steel grade of reinforcement for punching: A-IIIIN (B500SP); Characteristic strength = 500.00 MPa
- Bars perpendicular to edges



**Points where reinforcement is needed:**

Support no. / Point	Zone	Range: (m)		As (cm2) n x $\phi$ (mm)	
		L1	L2	As (cm2)	n x $\phi$ (mm)
S4	1	0.13	0.13	3.43	13 $\phi$ 6.0
	2	0.27	0.27	4.19	15 $\phi$ 6.0
S6	1	0.13	0.13	3.78	14 $\phi$ 6.0
	2	0.30	0.30	4.33	16 $\phi$ 6.0
S10	1	0.13	0.13	3.45	13 $\phi$ 6.0
	2	0.27	0.27	4.19	15 $\phi$ 6.0
S14	1	0.13	0.13	3.34	12 $\phi$ 6.0
	2	0.23	0.23	4.03	15 $\phi$ 6.0
S17	1	0.13	0.13	7.19	26 $\phi$ 6.0
	2	0.29	0.29	7.19	26 $\phi$ 6.0
S22	1	0.13	0.13	4.14	15 $\phi$ 6.0
	2	0.14	0.14	4.14	15 $\phi$ 6.0
S24	1	0.13	0.13	7.62	27 $\phi$ 6.0
	2	0.31	0.31	7.62	27 $\phi$ 6.0

S28	1	0.13	0.13	5.54	20 $\phi$ 6.0
	2	0.21	0.21	5.54	20 $\phi$ 6.0
S32	1	0.13	0.13	6.38	23 $\phi$ 6.0
	2	0.25	0.25	6.38	23 $\phi$ 6.0
S34	1	0.13	0.13	4.90	18 $\phi$ 6.0
	2	0.17	0.17	4.90	18 $\phi$ 6.0
S40	1	0.13	0.13	3.34	12 $\phi$ 6.0
	2	0.14	0.14	3.46	13 $\phi$ 6.0
S42	1	0.13	0.13	3.34	12 $\phi$ 6.0
	2	0.21	0.21	3.90	14 $\phi$ 6.0
S46	1	0.13	0.13	3.34	12 $\phi$ 6.0
	2	0.25	0.25	4.10	15 $\phi$ 6.0
S47	1	0.13	0.13	8.15	29 $\phi$ 6.0
	2	0.31	0.31	8.15	29 $\phi$ 6.0
	3	0.49	0.49	8.15	29 $\phi$ 6.0
	4	0.67	0.67	8.15	29 $\phi$ 6.0
	5	0.85	0.85	8.15	29 $\phi$ 6.0
	6	1.03	1.03	8.15	29 $\phi$ 6.0
S49	1	0.13	0.13	7.22	26 $\phi$ 6.0
	2	0.29	0.29	7.22	26 $\phi$ 6.0
	3	0.44	0.44	7.22	26 $\phi$ 6.0
	4	0.60	0.60	7.22	26 $\phi$ 6.0
	5	0.76	0.76	7.22	26 $\phi$ 6.0
	6	0.91	0.91	7.22	26 $\phi$ 6.0
S56	1	0.13	0.13	5.77	21 $\phi$ 6.0
	2	0.22	0.22	5.77	21 $\phi$ 6.0

**1.5.4. Deflection**  
 $|f(+)| = 0.0 \text{ (cm)} \leq f_{dop(+)} = 3.0 \text{ (cm)}$   
 $|f(-)| = 0.0 \text{ (cm)} \leq f_{dop(-)} = 3.0 \text{ (cm)}$

**1.5.5. Cracking**  
 upper layer  
 $a_x = 0.00 \text{ (mm)} \leq a_{dop} = 0.40 \text{ (mm)}$   
 $a_y = 0.00 \text{ (mm)} \leq a_{dop} = 0.40 \text{ (mm)}$   
 lower layer  
 $a_x = 0.00 \text{ (mm)} \leq a_{dop} = 0.40 \text{ (mm)}$   
 $a_y = 0.00 \text{ (mm)} \leq a_{dop} = 0.40 \text{ (mm)}$

### 3. Results - detailing

List of solutions:  
 Reinforcement: bars

Solution no.	Reinforcement range Diameter / Weight	Total weight (kG)
1	-	66392.11

Results for the solution no. 1  
 Reinforcement zones

Bottom reinforcement Name	coordinates				Provided reinforcement At $\phi$ (mm) / (cm)	
	x1 (cm <sup>2</sup> /m)	y1	x2 (cm <sup>2</sup> /m)	y2		
1/1-(1/3-) Ax Main <	-2.64 73.11	23.70	-1.75 25.30		32.0 / 11.0	66.26
1/2-(1/3-) Ax Main <	-2.63 73.11	-1.22	2.69	3.62	32.0 / 11.0	64.21
1/3- Ax Main <	-4.51 36.56	-6.51	33.08	28.55	32.0 / 22.0	15.85

1/4- Ay Perpendicular <	-4.51 33.51	-6.51	33.08	28.55	32.0 / 24.0	30.33
<b>Top reinforcement</b>						
<b>Name</b>	<b>coordinates</b>				<b>Provided reinforcement At</b>	
	<b>x1</b>	<b>y1</b>	<b>x2</b>	<b>y2</b>	<b>φ (mm) / (cm)</b>	
	<b>(cm2/m)</b>		<b>(cm2/m)</b>			
1/1+(1/14+) Ax Main <	-2.64 89.36	23.70	-0.88	25.30	32.0 / 9.0	30.24
1/2+(1/14+) Ax Main <	-2.63 89.36	0.00	-0.88	0.84	32.0 / 9.0	78.06
1/3+(1/14+) Ax Main <	4.50 89.36	-6.51	6.47	-4.73	32.0 / 9.0	24.19
1/4+(1/14+) Ax Main <	1.79 22.34	-6.51	8.39	-1.22	32.0 / 36.0	20.00
1/5+(1/14+) Ax Main <	-4.51 22.34	1.68	9.32	21.19	32.0 / 36.0	13.61
1/6+(1/14+) Ax Main	-2.64	21.19	0.00	25.30	32.0 / 36.0	8.82 <
1/7+(1/14+) Ax Main <	-2.63 22.34	0.00	9.32	1.68	32.0 / 36.0	18.87
1/8+(1/14+) Ax Main	0.00	21.19	33.08	22.09	32.0 / 36.0	5.02 <
1/9+(1/14+) Ax Main	0.48	-6.51	9.32	-4.73	32.0 / 36.0	5.02 <
1/10+(1/14+) Ax Main <	0.48 22.34	-1.22	9.32	0.00	32.0 / 36.0	12.19
1/11+(1/14+) Ax Main	1.79	-4.73	9.32	-1.22	32.0 / 36.0	5.02 <
1/12+(1/14+) Ax Main <	4.50 22.34	22.09	33.08	25.30	32.0 / 36.0	17.43
1/13+(1/14+) Ax Main	7.46	25.30	33.08	28.55	32.0 / 36.0	5.30 <
1/14+ Ax Main	9.32	13.35	33.08	21.19	32.0 / 36.0	6.26 <
1/15+(1/27+) Ay Perpendicular 21.0	-2.64 31.25 <	23.70	-0.88 38.30	25.30		32.0 /
1/16+(1/27+) Ay Perpendicular 21.0	-2.63 19.75 <	0.00	-0.88 38.30	1.68		32.0 /
1/17+(1/27+) Ay Perpendicular 42.0	4.50 12.76 <	-6.51	9.32 19.15	-2.97		32.0 /
1/18+(1/27+) Ay Perpendicular 42.0	-4.51 8.67 <	1.68	9.32 19.15	21.19		32.0 /
1/19+(1/27+) Ay Perpendicular 42.0	-2.64 9.15 <	21.19	0.00 19.15	25.30		32.0 /
1/20+(1/27+) Ay Perpendicular 42.0	-2.63 9.23 <	0.00	9.32 19.15	1.68		32.0 /
1/21+(1/27+) Ay Perpendicular 42.0	0.00 4.78 <	21.19	33.08 19.15	22.09		32.0 /
1/22+(1/27+) Ay Perpendicular 42.0	0.48 4.78 <	-6.51	9.32 19.15	-4.73		32.0 /
1/23+(1/27+) Ay Perpendicular 42.0	0.48 7.53 <	-1.22	9.32 19.15	0.00		32.0 /
1/24+(1/27+) Ay Perpendicular 42.0	1.79 4.78 <	-4.73	9.32 19.15	-1.22		32.0 /
1/25+(1/27+) Ay Perpendicular 42.0	4.50 13.50 <	22.09	33.08 19.15	25.30		32.0 /
1/26+(1/27+) Ay Perpendicular 42.0	7.46 4.78 <	25.30	33.08 19.15	28.55		32.0 /
1/27+ Ay Perpendicular	9.32	13.35	33.08	21.19	32.0 / 42.0	6.93 <

#### 4. Material survey

- Concrete volume = 223.37 (m<sup>3</sup>)
- Formwork = 744.55 (m<sup>2</sup>)
- Slab circumference = 154.22 (m)
- Area of openings = 0.00 (m<sup>2</sup>)
  
- Steel A-IIIIN (B500SP)
- Total weight = 70779.97 (kG)
- Density = 316.88 (kG/m<sup>3</sup>)
- Average diameter = 32.0 (mm)
- Survey according to diameters:

Diameter	Length (m)	Number:
32	1.65	7
32	1.72	8
32	1.86	4
32	2.10	4
32	2.58	37
32	2.70	5
32	2.87	6
32	4.27	14
32	5.81	8
32	6.64	9
32	7.32	170
32	7.47	26
32	8.01	312
32	8.19	16
32	8.62	8
32	8.73	13
32	8.78	9
32	8.85	34
32	8.96	48
32	9.18	24
32	9.28	17
32	9.35	14
32	9.39	24
32	9.62	24
32	9.78	14
32	9.82	114
32	9.94	86
32	10.16	26
32	10.25	114
32	10.38	43
32	11.02	38
32	11.45	25
32	11.89	13

### 6.4 Robot note for Foundation

#### 1 Spread footing: Foundation36

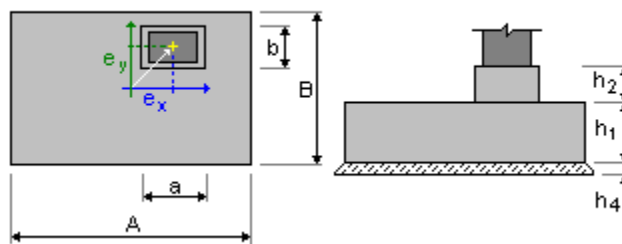
Number: 1

##### 1.1 Basic data

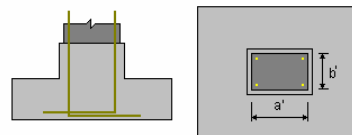
###### 1.1.1 Assumptions

- Geotechnic calculations according to : EN 1997-1:2008
- Concrete calculations according to : PN-EN 1992-1-1:2008
- Shape selection : without limits

###### 1.1.2 Geometry:



A	= 2.00 (m)	a	= 0.40 (m)		
B	= 2.00 (m)	b	= 0.40 (m)	$e_x$	= 0.00 (m)
h1	= 0.50 (m)	$e_y$	= -0.00 (m)		
h2	= 0.00 (m)				
h4	= 0.10 (m)				



a'	= 40.0 (cm)
b'	= 40.0 (cm)
cnom1	= 6.0 (cm)
cnom2	= 6.0 (cm)
Cover deviations: Cdev = 1.0(cm), Cdur = 0.0(cm)	

###### 1.1.3 Materials

- Concrete : C45/55; Characteristic strength = 45.00 MPa  
Unit weight = 2501.36 (kG/m3)  
Rectangular stress distribution [3.1.7(3)]
- Longitudinal reinforcement : type A-IIIN (B500SP)  
Characteristic strength = 500.00 MPa  
Ductility class: C  
Horizontal branch of the stress-strain diagram
- Transversal reinforcement : type A-IIIN (B500SP)  
Characteristic strength = 500.00 MPa
- Additional reinforcement: : type A-III (RB400W)  
Characteristic strength = 400.00 MPa

## 1.2 Geotechnical design

### 1.2.1 Assumptions

- Cohesion reduction coefficient: 0.00
- Smooth precast foundation 6.5.3(10)
- Sliding with soil pressure considered: for X and Y directions
- Design approach: 1  
A1 + M1 + R1
- $\gamma_{\phi'}$  = 1.00
- $\gamma_{c'}$  = 1.00
- $\gamma_{cu}$  = 1.00
- $\gamma_{qu}$  = 1.00
- $\gamma_{\gamma}$  = 1.00
- $\gamma_{R,v}$  = 1.00
- $\gamma_{R,h}$  = 1.00
- A2 + M2 + R1
- $\gamma_{\phi'}$  = 1.25
- $\gamma_{c'}$  = 1.25
- $\gamma_{cu}$  = 1.40
- $\gamma_{qu}$  = 1.40
- $\gamma_{\gamma}$  = 1.00
- $\gamma_{R,v}$  = 1.00
- $\gamma_{R,h}$  = 1.00

### 1.2.2 Soil:

Soil level:	$N_1$	= 0.00 (m)
Column pier level:	$N_a$	= 0.00 (m)
Minimum reference level:	$N_f$	= -0.50 (m)

#### Żwir gliniasty

- Soil level: 0.00 (m)
- Unit weight: 2243.38 (kG/m<sup>3</sup>)
- Unit weight of solid: 2702.25 (kG/m<sup>3</sup>)
- Internal friction angle: 18.3 (Deg)
- Cohesion: 0.03 (MPa)

### 1.2.3 Limit states

#### Stress calculations

	Soil type under foundation: not layered
	Design combination <b>ULS : KOMB1 N=964.07 M<sub>x</sub>=-4.44</b>
<b>M<sub>y</sub>=12.31 F<sub>x</sub>=2.50 F<sub>y</sub>=1.73</b>	
Load factors:	<b>1.35</b> * Foundation weight <b>1.35</b> * Soil weight
Calculation results: On the foundation level	
Weight of foundation and soil over it: Gr = 66.23 (kN)	
Design load:	

Nr = 1030.30 (kN)	Mx = -5.31 (kN*m)	My = 13.56 (kN*m)
Load eccentricity:		
eB = 0.01 (m)	eL = 0.01 (m)	
Equivalent foundation dimensions:		
B' = B - 2 eB  = 1.97 (m)		
L' = L - 2 eL  = 2.00 (m)		
Foundation depth:	Dmin = 0.50 (m)	
<b>Allowable stress calculation method: Semi-empirical - stress limit</b>		
qu = 0.30 (MPa)		
ple* = 0.29 (MPa)		
De = Dmin - d = 0.50 (m)		
kp = 1.00		
q'o = 0.01 (MPa)		
qu = kp * (ple*) + q'o = 0.30 (MPa)		
Stress in soil: qref = 0.27 (MPa)		
Safety factor: qlim / qref = 1.104 > 1		
<b>Uplift</b>		
<u>Uplift in ULS</u>		
Design combination	<b>ULS : KOMB15 N=843.66 Mx=-17.17</b>	
<b>My=-12.28 Fx=-1.78 Fy=4.23</b>		
Load factors:	<b>1.00</b> * Foundation weight	
	<b>1.00</b> * Soil weight	
Contact area:	s = 0.02	
	slim = 0.17	
<b>Sliding</b>		
Design combination	<b>ULS : COMB19 N=-0.22 Mx=0.11 My=-</b>	
<b>0.09 Fx=-0.02 Fy=-0.02</b>		
Load factors:	<b>1.00</b> * Foundation weight	
	<b>1.00</b> * Soil weight	
Weight of foundation and soil over it:	Gr = 49.06 (kN)	
Design load:		
Nr = 48.84 (kN)	Mx = 0.12 (kN*m)	My = -0.10 (kN*m)
Equivalent foundation dimensions: A_ = 2.00 (m) B_ = 2.00 (m)		
Sliding area:	4.00 (m <sup>2</sup> )	
Foundation/soil friction coefficient:	tan(δd) = 0.19	
Cohesion:	cu = 0.03 (MPa)	
Soil pressure considered:		
Hx = -0.02 (kN)	Hy = -0.02 (kN)	
Ppx = 5.27 (kN)	Ppy = 5.27 (kN)	
Pax = -1.43 (kN)	Pay = -1.43 (kN)	
Sliding force value	Hd = 0.00 (kN)	
Value of force preventing foundation sliding:		
- On the foundation level:	Rd = 9.20 (kN)	
Stability for sliding:	∞	
<b>Average settlement</b>		



Soil type under foundation: not layered  
 Design combination **SLS : COMB18 N=678.91 Mx=-3.42**  
**My=4.65 Fx=1.09 Fy=1.21**  
 Load factors: **1.00** \* Foundation weight  
**1.00** \* Soil weight  
 Weight of foundation and soil over it: Gr = 49.06 (kN)  
 Average stress caused by design load: q = 0.18 (MPa)  
 Thickness of the actively settling soil: z = 5.00 (m)  
 Stress on the level z:  
 - Additional:  $\sigma_{zd} = 0.01$  (MPa)  
 - Caused by soil weight:  $\sigma_{z\gamma} = 0.12$  (MPa)  
 Settlement:  
 - Original  $s' = 0.7$  (cm)  
 - Secondary  $s'' = 0.0$  (cm)  
 - TOTAL  $S = 0.7$  (cm) < Sadm = 5.0 (cm)  
 Safety factor: 7.149 > 1

#### Settlement difference

Design combination **SLS : COMB18 N=678.91 Mx=-3.42**  
**My=4.65 Fx=1.09 Fy=1.21**  
 Load factors: **1.00** \* Foundation weight  
**1.00** \* Soil weight  
 Settlement difference: S = 0.0 (cm) < Sadm = 5.0 (cm)  
 Safety factor: 156.9 > 1

#### Rotation

About OX axis  
 Design combination **ULS : KOMB15 N=843.66 Mx=-17.17**  
**My=-12.28 Fx=-1.78 Fy=4.23**  
 Load factors: **1.00** \* Foundation weight  
**1.00** \* Soil weight  
 Weight of foundation and soil over it: Gr = 49.06 (kN)  
 Design load:  
 Nr = 892.72 (kN) Mx = -19.29 (kN\*m) My = -13.17 (kN\*m)  
 Stability moment: Mstab = 892.72 (kN\*m)  
 Rotation moment: Mrenv = 19.29 (kN\*m)  
 Stability for rotation: 46.28 > 1

About OY axis  
 Design combination: **ULS : KOMB13 N=850.00 Mx=-3.58**  
**My=15.36 Fx=2.96 Fy=1.54**  
 Load factors: **1.00** \* Foundation weight  
**1.00** \* Soil weight  
 Weight of foundation and soil over it: Gr = 49.06 (kN)  
 Design load:  
 Nr = 899.06 (kN) Mx = -4.35 (kN\*m) My = 16.84 (kN\*m)  
 Stability moment: Mstab = 899.06 (kN\*m)  
 Rotation moment: Mrenv = 16.84 (kN\*m)  
 Stability for rotation: 53.38 > 1

### 1.3 RC design

### 1.3.1 Assumptions

- Exposure : XF2
- Structure class : S4

### 1.3.2 Analysis of punching and shear

#### Punching

Design combination	<b>ULS : KOMB1 N=964.07 Mx=-4.44 My=12.31</b>
<b>Fx=2.50 Fy=1.73</b>	
Load factors:	<b>1.35 * Foundation weight</b> <b>1.35 * Soil weight</b>
Design load:	
Nr = 1030.30 (kN)	Mx = -5.31 (kN*m)      My = 13.56 (kN*m)
Length of critical circumference:	3.76 (m)
Punching force:	704.27 (kN)
Section effective height	heff = 0.43 (m)
Reinforcement ratio:	$\rho = 0.20 \%$
Shear stress:	0.55 (MPa)
Admissible shear stress:	1.28 (MPa)
Safety factor:	2.346 > 1

### 1.3.3 Required reinforcement

#### Spread footing:

bottom:

ULS : KOMB1 N=964.07 Mx=-4.44 My=12.31 Fx=2.50 Fy=1.73  
My = 183.64 (kN\*m)       $A_{sx} = 8.50 \text{ (cm}^2\text{/m)}$

ULS : KOMB3 N=960.27 Mx=-12.59 My=-4.28 Fx=-0.34 Fy=3.35  
Mx = 183.21 (kN\*m)       $A_{sy} = 8.50 \text{ (cm}^2\text{/m)}$

$A_{s \text{ min}} = 8.50 \text{ (cm}^2\text{/m)}$

top:

ULS : COMB19 N=-0.22 Mx=0.11 My=-0.09 Fx=-0.02 Fy=-0.02  
My = -0.07 (kN\*m)       $A'_{sx} = 8.50 \text{ (cm}^2\text{/m)}$

ULS : COMB19 N=-0.22 Mx=0.11 My=-0.09 Fx=-0.02 Fy=-0.02  
Mx = -0.07 (kN\*m)       $A'_{sy} = 8.50 \text{ (cm}^2\text{/m)}$

$A_{s \text{ min}} = 8.50 \text{ (cm}^2\text{/m)}$

#### Column pier:

Longitudinal reinforcement      A = 0.00 (cm<sup>2</sup>)       $A_{\text{min.}} = 0.00 \text{ (cm}^2\text{)}$   
A = 2 \* (Asx + Asy)  
Asx = 0.00 (cm<sup>2</sup>)      Asy = 0.00 (cm<sup>2</sup>)

### 1.3.4 Provided reinforcement

#### Spread footing:

**Bottom:**

Along X axis:

16 A-IIIN (B500SP) 12      l = 1.88 (m)      e = 1\*-0.89 + 15\*0.12

Along Y axis:

16 A-IIIN (B500SP) 12      l = 1.88 (m)      e = 1\*-0.89 + 15\*0.12

**Top:**

Along X axis:

16 A-IIIN (B500SP) 12      l = 1.88 (m)      e = 1\*-0.89 + 15\*0.12

Along Y axis:

16 A-IIIN (B500SP) 12      l = 1.88 (m)      e = 1\*-0.89 + 15\*0.12

**Pier**

**Longitudinal reinforcement**

Along Y axis:

4 A-IIIN (B500SP) 12      l = 0.50 (m)      e = 1\*-0.13 + 1\*0.25

**Transversal reinforcement**

3 A-IIIN (B500SP) 8      l = 1.22 (m)      e = 1\*0.25 + 2\*0.09

3 A-IIIN (B500SP) 8      l = 1.31 (m)      e = 1\*0.56 + 2\*0.10

**Dowels**

**Longitudinal reinforcement**

4 A-IIIN (B500SP) 12      l = 0.83 (m)      e = 1\*-0.14 + 1\*0.28

**2 Material survey:**

- Concrete volume                      = 2.00 (m3)
- Formwork                                = 4.00 (m2)
- Steel A-IIIN (B500SP)
  - Total weight                            = 114.58 (kG)
  - Density                                    = 57.29 (kG/m3)
  - Average diameter                      = 11.8 (mm)
  - Survey according to diameters:

Diameter	Length (m)	Number:
8	1.22	3
8	1.31	3
12	0.50	4
12	0.83	4
12	1.88	64

---

## 7 References

- PN-EN 1992-1-1 (2004) : Eurocode 2: Design of concrete structures - Part 1-1: General rules and rules for buildings.
- PN-EN 1992-1-2 (2004) : Eurocode 2: Design of concrete structures - Part 1-2: General rules - Structural fire design.
- PN-EN 1991-1-1 (2004) : Eurocode 1: Actions On Structures - Part 1-1: General Actions - Densities, self-weight, imposed loads for buildings.
- PN-EN 1991-1-3 (2008) : Eurocode 1: Actions On Structures - Part 1-3: General Actions - Snow loads.
- PN-EN 1991-1-4 (2008) : Eurocode 1: Actions On Structures - Part 1-4: General Actions - Wind Actions.
- Adam Zybura " Konstrukcje żelbetowe - Atlas rysunków"
- Włodzimierz Starosolski ( Warszawa 2007) " Konstrukcje żelbetowe według PN-B-03264:2002 i Eurokodu2"
- Salman Azhar, Ph.D. ( July 2011) " Building Information Modeling (BIM): Trends, Benefits, Risks, and Challenges for the AEC Industry"
- David Bryde, Marti Broquetas, Jurgen Marc Volm ( December 2012) "The project benefits of Building Information Modelling (BIM)"
- Autodesk (April 2010) Revit Architecture 2011 User's Guide
- <http://www.autodesk.com/solutions/bim/buildings/structures-and-engineering>
- [https://en.wikipedia.org/wiki/Building\\_information\\_modeling](https://en.wikipedia.org/wiki/Building_information_modeling)
- <https://bimopedia.com/2013/04/02/5-tips-to-reduce-bad-performance-in-revit/>
- <http://www.solibri.com/support/bim-ifc/>
- <http://www.buildingsmart-tech.org/specifications/ifc-overview>