Dotto Mkinga

Prota Structure Modelling Reinforced Concrete Members for a Single Family Detached House

Metropolia University of Applied Sciences

Bachelor of Civil Engineering

Sustainable Building Engineering

Thesis

7 February 2019

Author Title Number of Pages Date	Dotto Mkinga Prota Structure Modelling Reinforced Concrete Members for a Single Family Detached House 43 pages + 1 appendix 7 February 2019
Degree	Bachelor of Civil Engineering
Degree Programme	Civil Engineering
Professional Major	Sustainable Building Engineering
Thesis Supervisor	Sunil Suwal, Senior Lecturer

The aim of this Bachelor's was to model reinforced concrete members for a single-family detached home using Prota Structure. The paper also revealed how beams, columns, slabs and mat foundation were successfully placed and analyzed using different tools and features available on Prota Structure.

Furthermore, there were two methodologies used in designing the detached family home. The first approach was to model the architectural drawings with an ArchiCAD application and the second approach was to use Prota Structure in modelling, designing and analyzing reinforced concrete members. The latter approach was extensively explained in this paper as this was the main target. In addition, all reinforced concrete structural members were successfully executed with no errors, the checking for Axial Load Comparison Report, and other essential checks were successfully passed as expected using Eurocode standards.

The study also briefly explained the information exchange possibilities between Prota Structure and Autodesk Revit using a custom link called bi-directional link. This information exchange uses the Prota BIM ribbon available on Autodesk Revit and essentially can synchronize and track two applications effectively. More research is needed about the Prota Structure application, and this paper can be used as a suggestion.

Keywords	Prota Structure, single family detached home, ArchiCAD, structural engineers

Metropolia
University of Applied Sciences

Contents

List of Abbreviations

BIM   Building Information Modelling

FE   Finite Element

RC   Reinforced Concrete

DXF   Drawing Exchange Format

DPM   Damp Proof Membrane

As(d)-bot   Area of Steel diameter at the bottom

As(d)-top   Area of Steel diameter at the top

1 Introduction

Structural Engineering is a branch of Civil Engineering which deals with the design, construction and maintenance of our surrounding infrastructure for example buildings, bridges and tunnels. The people who are responsible for designing these tasks are called structural engineers. Meaning, their main task is to create the structural system or skeleton, which in turn physically supports the intended loads. All in all, the final goal of structural design is to resist loads such as gravitational, seismic and wind loads. [1.]

Today, the field of structural engineering has enormously benefited from the number of computer applications that have been developed. Computer applications can be utilized in any stage of the structural engineering process, including drafting of proposals and construction drawings and structural analysis tasks together with designing. Despite the fact that most structural engineering computer applications are more expensive, they offer benefits and flexibility in many ways. The major benefits of using computer applications in structural engineering might include huge productivity, increasing accuracy of analysis, design procedures, design optimization, more economical designs, improved record keeping, file sharing, automation of calculations, running of complex calculations and analysis, and finally accurate modeling of complex structures. More specifically, there are different software applications available in the field of structure engineering. In this report, Prota Structure is being studied. The focus is on studying the workflow and benefits it can bring to structural engineers. [1.]

This thesis consists of eight chapters. The first chapter defines the term structural engineering and briefly explains some benefits of using computer software in the field of structural engineering. The second chapter focuses on a structural application called Prota Structure. This application aims to offer structural engineers various benefits when designing concrete and steel buildings, including the possibilities of modeling and analyzing the models.

The third chapter starts by defining a single-family detached house. In addition, it lists the key characteristics of designing the floor plan of single-family detached house. Furthermore. the chapter explains the key stages that need to be considered before designing. Chapter four explains how the architectural design is executed. Chapter five

explains the modeling structural members such as axes, beams, columns, slabs, wall loads, as well as the design of raft foundation in more detail.

The sixth chapter describes the results of a building analysis, where various forces and moments were analyzed and examined. It also discusses the reinforcement design for beams, columns and slabs. Chapter seven, explains Prota BIM, which uses a bi-directional link between two applications called Prota Structure and Autodesk Revit for exchanging information and modifying data. Finally, chapter eight is a conclusion of what has been studied.

2 Prota Structure

Prota Structure is a powerful application that helps engineers in modelling, analyzing and designing steel and concrete buildings quickly and accurately. Prota Structure is owned by a Turkish company called Prota Engineering. The company was established in 1985 and of few years after its inception, it turned out to be one of the country's leading engineering and consultancy firms, specializing in the designing of both steel and concrete buildings, using extensive ranges of disciplines. Prota Engineering operates within strictly technical institutional structure that consists of different experts as partners. [2.]

The application is known to be fast in delivering projects with full consolidated concrete and steel design using a single central model. It is also accompanied by multiple window views and dynamic input which makes the modelling tasks easy, quick and intuitive. In addition, it is a state-of-the art structural model using 3D finite element analysis with numerous analysis preferences and shell element support for storey and shear walls. These analysis options include vertical and lateral elastic analysis, equivalent static earthquake load, response spectrum analysis, time-history, pushover, staged construction, P-delta, temperature difference and seismic isolators. [2.]

 Apart from using various globally recognized codes, including specialist seismic requirements for analyzing and designing buildings, it also offers fully automated RC detailing in drawing sheets with its dynamic quantity tables along with fast engineering macros including retaining wall, stair, pile analysis and design and detailing. It also has custom link for RC and Steel models with full-fledged BIM platforms such as Autodesk

and Tekla structure. [2.] The benefits of Prota Structure are significant for structural designers and engineers. The main benefits that Prota Structure provides are listed in table 1. [2.]

Table 1. Main benefits of Prota Structure [2.]

Benefits of protastructure	Description
One central model	- model from simple to complex buildings with speed, ease and accurate information.
Seamless Documentation	-clear, consistent and concise drawings -high quality creation with ProtaDetails and ProtaSteel
Intelligent physical objects	- models can be accurately and directly shared with other BIM platforms, including Autodesk Revit.
Leading BIM Integration	-Custom links for Reinforced Concrete and Steel structure with well-known BIM platforms including Autodesk Revit and Tekla Structures. -it saves hours of modelling time plus smoothly making changes and enhancements and keeping the BIM model up to date.
Automated Loading	-time-saving with auto-generated loading that include yield line, finite element, wind and seismic loading combined with automated load combination generation in accord to the chosen code of practice
Seismic Assessment	-involves the use of well-advanced features such as pushover analysis for assessing and rehabilitating existing structures
Manages Changes	Prota Structure can easily deal with any changes and quickly, rectifying and update them whenever is required during the project execution.
Modelling	-easy and quick modeling capabilities -models can be merged and project can be created and modified easily - interface and tools are effortlessly understandable
Analysis	-very modern and uses the latest ideas and methods for 3D finite element solver and analytical model -slab analysis systems can be done independently or combined with the structure as a single entity by using finite element. -it has full-featured analysis post-processor which helps to visualize features such as stress contours, deformations, forces and moment diagrams in real-time.
Design and Foundations	-compatible with well-known code-based design for steel components such as Eurocode 3, BS5950, AISC360-10 (ASD, LRFD) and new Turkish steel design codes. -it has wide range of features and functions for seismic analysis and uses design codes based on Eurocode 8 and UBC and IBC and TEC. -it offers automatic reinforced concrete design by using different codes such as ACI1318, Eurocode 2, BS810-1997, CP65, HK2004, BS6399, TS498 and TS500. -advanced documentation tools for generating design report. -it uses both analytical model and finite element methods to design all types of foundations
Prota details	-it needs only single click to produce drawing sheet details by using self-regulating mechanism. -it can create dynamic quantity tables, which means whenever the changes occur the tables and bar bend schedules will be updated immediately. - automatically update any changes from prota structure as they occur.
Prota steel	-Prota structure and Prota steel can be easily communicated and so with help of central 3D model, it saves time for self-activating connections design and the self-activating preparation of detail drawings. -it has fully-featured parametric connection library and as a result modelling and steel design connections can be smoothly performed. - automatically identifying clashes and conflicts between the elements.

Table 1 describes the main benefits of Prota Structure. These benefits make the whole task of creating the model smooth, efficient and effective. Most importantly, time can be easily managed throughout designing process and project delivery, and thus offering a greater productivity and profitability. [2.]

3   Project Description

By definition, a single-family detached house is a structure maintained and used as single dwelling unit. There are following characteristics in which dwelling unit possess:

- No common walls; this type of home is a stand-alone detached property. Meaning, it is a home that does not share common walls or a roof with any other dwelling.
- Land; a single-family home has no shared property but is constructed within its own designated area.
- Utilities; must consist of only one set of utilities. On many occasions, sharing these utilities with another residence may not be possible. This includes heating, electricity, water and other essential services.
- Entrance and exit; Unlike apartments, a single-family house must have its own private and direct access to a street or thoroughfare.
- One owner; it must be built as a domicile for one family, person, or household, whose homeowner has an undivided interest in the property.
- Single kitchen; a single-family home should have one kitchen. [3.]

As the main theme was to study the Prota Structure and its workflow for designing a single family detached house's concrete members. Nevertheless, both architectural concept and structural concept had to be initiated before drawings for the design of structural concrete members were prepared. This was very important, because it helps structural engineer as to how to place and analyze concrete members or concrete elements in accordance to architect's proposal. [4.]

The application used for architectural design was ArchiCAD, one of the powerful tools in designing architectural elements. The analyze and design concrete members for structural drawings on the other hand was executed by Prota Structure.

First and foremost, before designing a single-detached family house floor plan, there are key design stages that need to be considered. These stages are listed on table 2 below. [4.]

Table 2. Key design stages [4.]

Key stages	Description
Developing project in brief	- architect and client discuss the client`s needs - client gives budget, time frame and other parameters that have an impact on the design process.
Designing of project	-expertise and going through all design stages -architect pins down and analyses four profound stages according to the client requirements; these requirements include; -Initial design→ presenting brief requirement and project proposal, -developed design → establishing layout of spaces, construction materials, consulting with other experts. -detailed design → establishing all necessary construction drawings, technical specifications with collaboration with other professionals. -tendering for a contractor→ preparing tender document for different contractors
Supervising the construction process	-architect will have a mandate to protect the client`s interest and also ensure all contracted tasks have been executed according to the contract documentation.

Table 2 outlines key design stages of any building project. In fact, there are flexibilities with these designing stages which in turn, makes the duration and complexity of each design stages to depend on the size and complexity of the building project involved. All in all, the goal is to establish clear expectations of each designing stages with the designing team and outline them in the agreement to avoid the future conflicts. [4.]

Architectural design and development are done by architects and structural engineers take the design as reference and prepare structural design for the project. [4.]The structural engineers will design all structural members using standard codes such as eurocodes and performing checks. The standard codes used for this final year project were Eurocode 1 (EN 1991) Actions on Structures (Base Code), and Eurocode 2 (EN 1992) Design of Concrete Structures (Base Code).

4 Architectural Design

The single-family detached house in this thesis is an imaginary project. It is a single-family house intended for a family of five occupants. The location of the project is assumed to be near the coast, and therefore the conditions are wet and soft.

The building has two floors, of which the ground floor consists of living room and, kitchen garage, two bedrooms, sauna, toilet and storage room. These spaces can be seen in figure 1. Also, as shown in figure 2, the first floor has a master bed room, a smaller bed room, gaming room, toilet, terrace and penthouse.
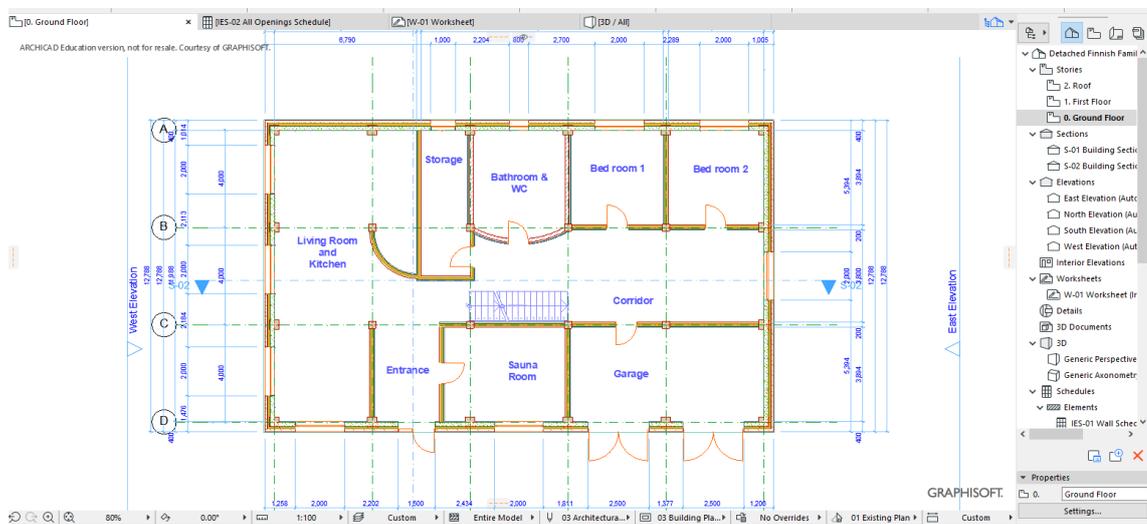
Figure 1. Ground Floor Plan View as displayed on ArchiCAD

The area of the building used in the thesis is 20700 mm long and 12700 mm wide, making the total area to be 262.9 m^2. The height of the walls on each floor was 3000 mm.
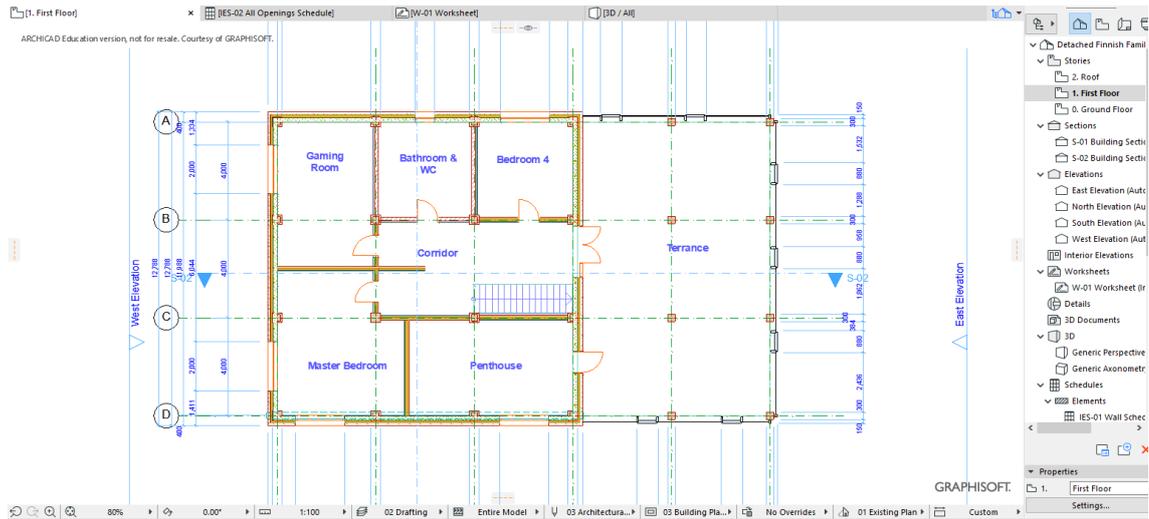
Figure 2. Second Floor Plan View as shown on ArchiCAD File

Figure 1 and Figure 2 display the plan views of both the ground floor and the first floor. These plan views give an insight of how the spaces were arranged, and they also show other features and components such as dimensions details, grids, walls, doors, windows, columns, stair, and curtain wall.

4.1   Raft Foundation

Since the designated area for the project has low bearing capacity, the use of raft foundation is inevitable. A raft foundation distributes the total load of the building on the entire ground floor area. The contact area of the raft foundation with soil is larger than that of any other type of foundation, so the load is spread evenly in a large area, and therefore, the soil will have a minimum stress, and the likelihood of shear failure of soil will consequently be minimized. [5.]

The raft foundation required for the building in the thesis was designed using the slab tool in ArchiCAD. A 400 mm thick slab with reinforced concrete material was selected at this stage in the architectural design to convey the architect`s viewpoint towards the use of raft foundation. [6.] The detailed analysis and modelling of the foundation is discussed in a structural model.

4.2 Exterior Wall

A composite wall of 400 mm was selected to be used in the building as the external envelope. As this is not a real project, it was assumed that the composite wall is easily available for the proposed location. Furthermore, composite wall with different layers of material composition as shown in figure 3 was also selected to see how the design application supported in the creation of such elements required in the design. [6.]
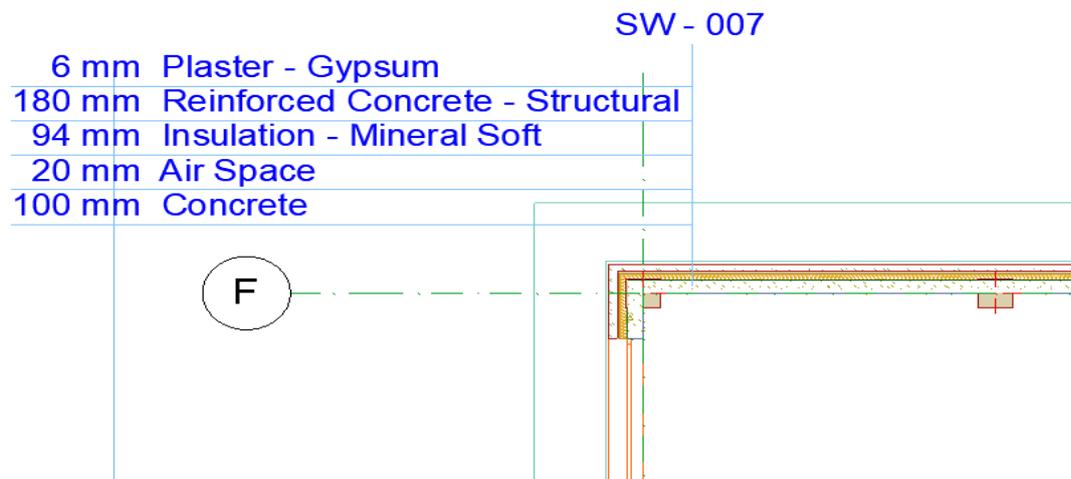

Figure 3. Exterior wall sandwich taken from ArchiCAD File. [6.]

Furthermore, other components of the building elements were used in a similar way with the specific tools meant for them. Wall elements were created with the wall tool, slab elements with the slab tools, columns with the column tool, steel stair with the stair tool and windows and doors with the window tool and door tool, respectively. This was done because of the existing BIM guidelines that state that a building element should be modeled with the right tools so that the information required for the elements are in the elements by default. Other components used in the design for the single-family detached home are listed in table 3. [10.]

Table 3. Components of the building [10.]

Component type	Details	Structure
Interior wall (for partition)	200 mm	Composite wall with -40 mm gypsum plaster, 100 mm insulation, 20 mm airspace and 40 mm gypsum plasterboard
Interior wall (for toilets)	230 mm	Composite wall with -40 mm gypsum plasterboard waterproof,100 mm insulation mineral soft, 20 mm airspace and 40 mm gypsum plasterboard waterproof
Terrance curtain wall material	Generic type for boundary frame sizes.	Curtain wall material - Metal Aluminum Frame, Glass Blue and Generic. a= frame width 100 mm, b=frame depth 300 mm, h=frame back`s offset 250 mm, d=depth of panel 30 mm and w=width of panel 20 mm
Flat roofing	250 mm	Composite slab with -4 mm Membrane Waterproof ,40 mm Insulation Fiber Hard, 10 mm Membrane-Vapor Barrier, 46 mm Concrete-Structural and 150 mm Reinforced Concrete including 25 mm concrete cover
Intermediate slab material (first floor slab)	200 mm	Composite slab with -200 mm reinforced concrete including 25 mm concrete cover
Main windows		Main windows dimensions are; -2000 mm width,1500 mm height and 1000 mm sill to story height
Bathroom and sauna windows		bathroom and sauna windows dimensions are; -800 mm width and 800 mm width
Main door		main doors dimensions are; -1500 mm width, 2100 mm height and 5 mm threshold
Internal doors		internal doors dimensions are; -900 mm width,2100 mm height and 5 mm threshold
Garage door		garage doors dimensions are; -2500 mm width2100 mm height and 5 mm threshold
Steel stair		Steel stair details are; -4000 mm flight length, 1200 mm width, 3000 mm stair width, 18 No. of risers, 222 mm tread depth and 167 mm tread height
Raft foundation	480 mm	-400 mm reinforced concrete, 4 mm water proof membrane, 74 mm thick unreinforced concrete and 2 mm timber floor finish
Columns		400 mm width and 400 mm length (square columns)

Table 3 shows all building components used for modeling the project [10]. These components are displayed, along with their dimensions and details. Indeed, some of these building components, such as raft foundation, walls and columns, are the structural

members, and therefore, the phenomenon of designing these members is clearly explained in the chapter 5.

5   Structural Design

Structural design is the methodical investigation of the stability, strength and rigidity of structures. The key objective in structural analysis and design is to produce a structure capable of preventing all applied loads without failure during its intended life. [1.] To model, analyze and design concrete members, Prota Structure uses the process that is described in the below figure 4 flow chart to achieve the intended task in question.

1. Build the Model	tabulated code coefficients

2. Derive Beam Loads using the Yield Line (tributary area) approach.	2a- FE Load Decomposition Derive Beam Loads by meshing slabs in FE model →Choose whether to use these loads selectively or on all beams

3. Run the general building analysis to generate column, wall and beam design forces.	3a- FE Floor Analysis (Flat Slab/Sub-frame) Use sequential FE-floor analyses to chase gravity loads down the structure → Merge member forces with lateral results from Building

3b Use same FE models to generate alternative slab design forces

4.1 Beam Design	4.2 Column/Wall	4.3 Foundation Design	4.4 Slab Design

Figure 4. Prota Structure design flow chart

5.1   The Process of Structural Design

The first step is to build a model. Secondly, after build a model, the beam loads are automatically derived using yield line/ tributary area approach. Thirdly, general building analysis can then be run to derive the column, wall and beam design forces. Fourthly, after running the general building analysis, the beam, column/wall and foundation design can be designed. [7.]

Alternatively, there is method to yield line decomposition from the derive a beam loads. The method is called as Finite Element Load Decomposition. It uses yield lines to derive beam loads by meshing slabs in the Finite Element model. Loads can then be chosen to be used selectively, or on all beams in the model, or in the current storey. [7.]

Moreover, there is also an alternative method to run a general building analysis. The method is called Finite Element Floor Analysis (Flat Slab/Sub-frame). This method is mandatory for a flat slab model. There is an optional method for beam slab model where the Sub-frame or single storey behavior is preferred. As a result, the Finite Element Floor Analysis method of running a general building analysis employs a sequential Finite Element floor analysis to chase down gravity loads from the roof down to the foundation. Beam and column forces can then be merged with the lateral results from the building analysis. Once the merging of forces with lateral results is done, all concrete members such as beams, column and wall, and the foundation can be performed and designed. [7.]

Finally, for the slab design, after building the model, a slab can be designed with a slab design based on the tabulated code coefficients of the yield line method. The same FE Floor analysis can be used to generate alternative slab design forces for the slab design. [7.]

5.2   Start-up Prota Structure and Opening Project Dialog

When opening the Prota Structure application, the initial window offers the possibilities to open either an existing project or a new project. When selecting an existing project, the programme provides the information from the project database, such as the project location, the version used to create the project, the user who created it, related material

grades, codes of practice, and number of stores. [7, p.6.] This information is essential because it provides the designer with an access to the information. Figure 5 highlights the process of opening the project.
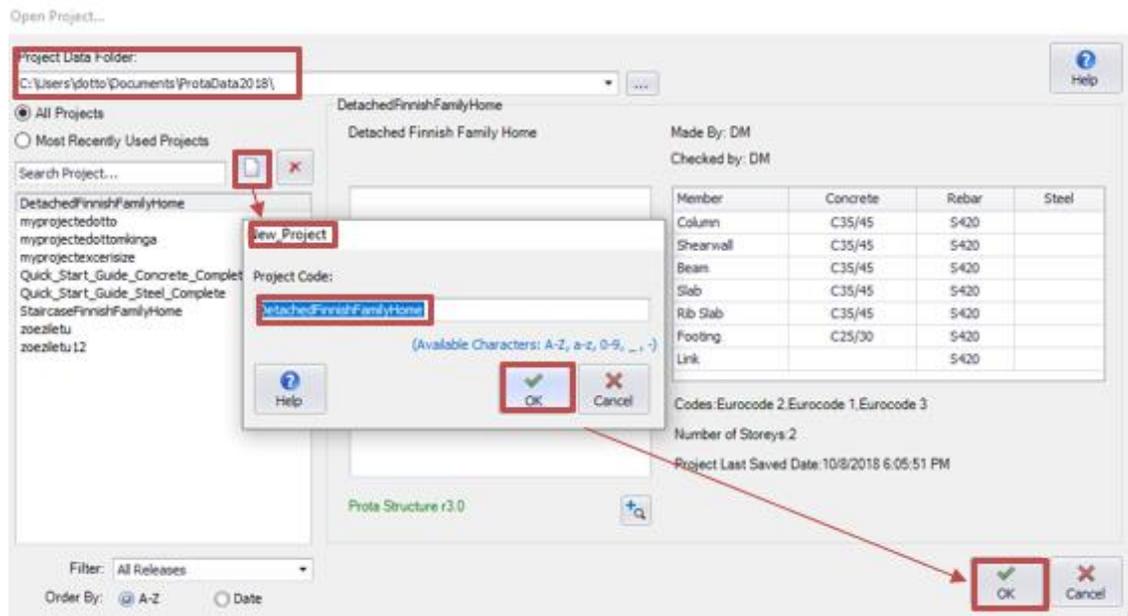

Figure 5. Startup of new project as it displayed on Prota Structure [7].

After opening a new project, the Setting Control Center will appear with pre-set templates. The templates are usually helpful in establishing different design parameters, including the design codes, the quality of different materials, design settings for members, et al. On the other hand, the template settings could also be extracted directly from the existing project. [7, p.7.]

The design codes for the project used in the thesis were Eurocode 1, 2 and 3. Therefore, IMPORT was selected to get to the User Interface as pointed out in figure 6.
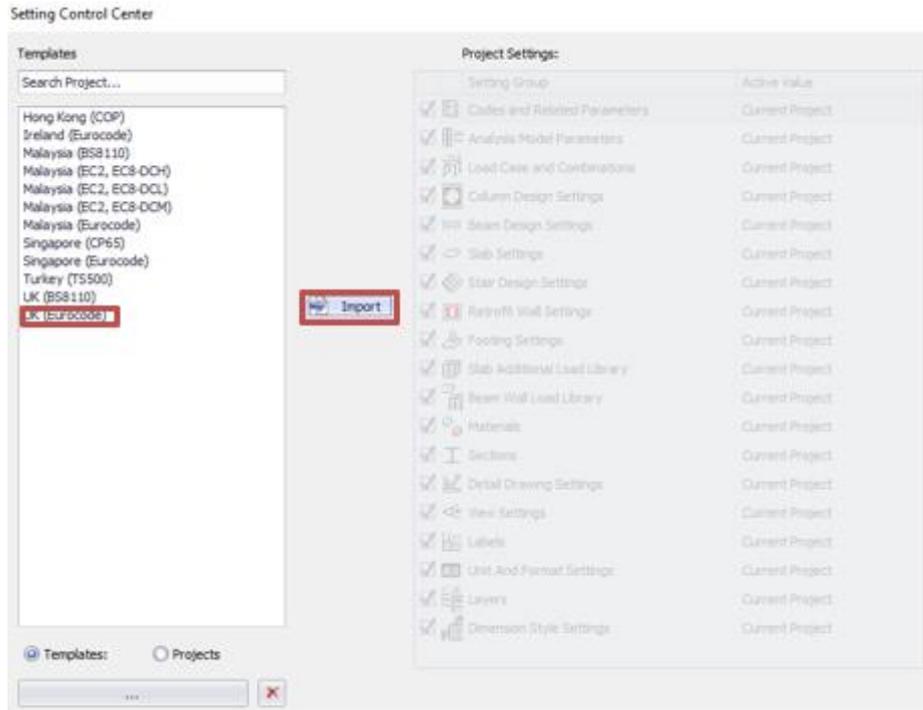
Figure 6. Importing template for Eurocode-based design [7].

The user interface of Prota Structure has various components such as file tool bars, edit tool bars, reinforced concrete (RC) tool bars, steel members tool bars, analysis & design tool bars and building set out tool bars. Also, the user interface contains structure tree tool bars, view tool bars, layers tool bars, plan view tool bars, 3D view tool bars, active storey level tool bars, units tool bars and finally cursor coordinates tool bars. [7, pp.3-11.]

Furthermore, the user interface combines the concrete and steel members modeling and designing them discretely, using either top dropdown menu or member toolbars icons. The same information that can be seen from concrete member dropdown menu can also be seen on the RC members tool bars. Figure 7 below shows the user interface in a big picture. [7, pp.3-11.]
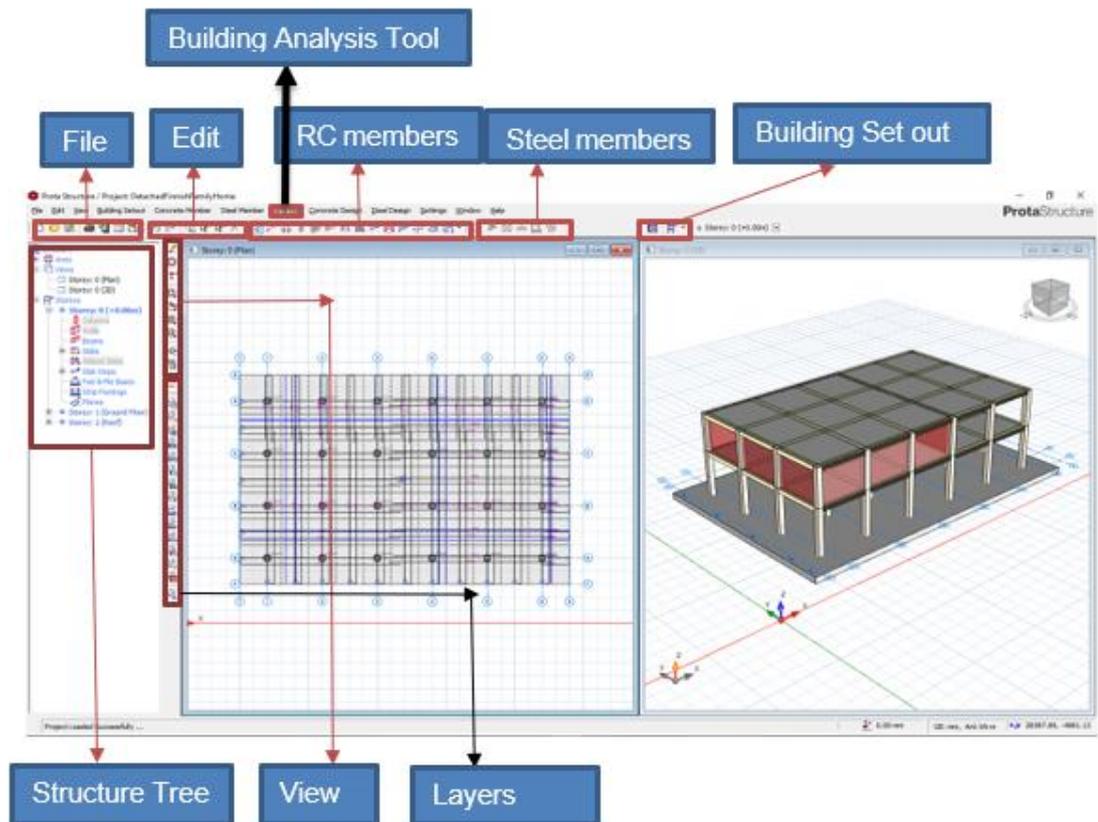
Figure 7. Prota Structure user interface [7.]

Figure 7 discloses Prota Structure user interface [7.]. The user interface displays all essential components required when modeling the final year project. These components include file, edit, building analysis tool, reinforced concrete members, steel members, and building set out. Other components are structure tree, view as well as layers. [7, p.3.]

5.3   External Reference Drawing Function and Creation of Members

In this final year project, the DXF (drawing exchange format) method was chosen to generate the axes references. The generation of axes was performed with the following detailed information: the Dir-1 Axes (horizontal axes) consisted three number of grids with 4000 mm distance each. The Dir-2 Axes (vertical axes) had five grids with 4000 mm distance each. In addition, offsetting axes both horizontally and vertically on both ends by 2000 mm was also executed for creating raft foundation. As can be seen in figure 8, the DXF geometry reference was imported to a Prota Structure. [7, pp.12-13.]
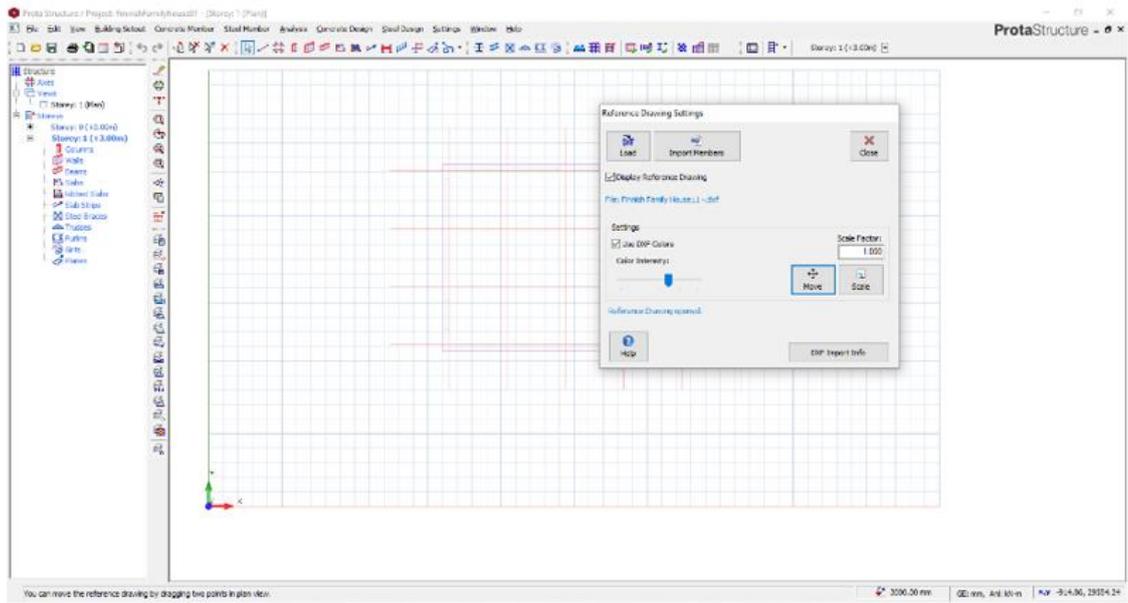
Figure 8. DXF geometry reference into a Prota Structure [7, pp.12-13.]

Creation of Columns

Columns for the project in this thesis were created with column icon. After clicking the column tool, the column properties will pop-up. The abbreviation b1 is the width of the column and b2 is the height of the column. Columns were inserted at their respective intersection points called column reference point. The size of the all Reinforced Concrete columns (b1 and b2) was 400 mmx400 mm and both e1 and e2 (eccentricity value) were given the value zero. This means that, all columns were placed centrally as it can be seen in figure 9. [7, pp.14-15.]
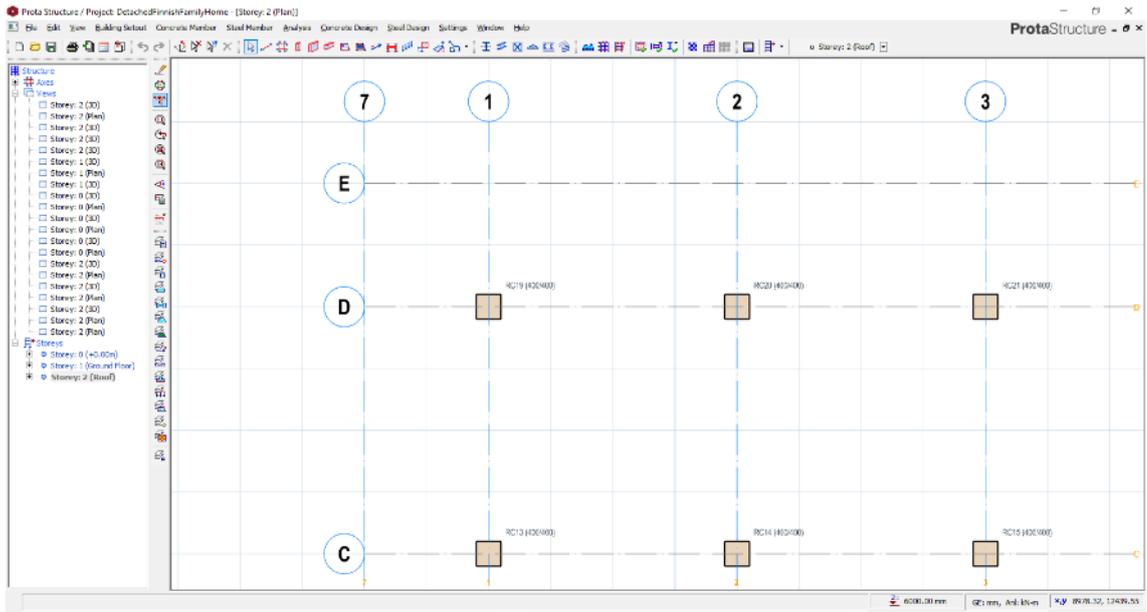
Figure 9. Column Creation as it is showed on Prota Structure File. [7.]

Creation of Beams

The beams for the detached in this thesis were created by using the beam tool bar. Beam creation uses a similar approach as wall creation. This means that a beam requires two insertion points called I and J, of which I is the left end (horizontal) or bottom end (vertical) and J is the right end (horizontal) or upper end (vertical). Figure 10 shows all parameters needed for beam creation.
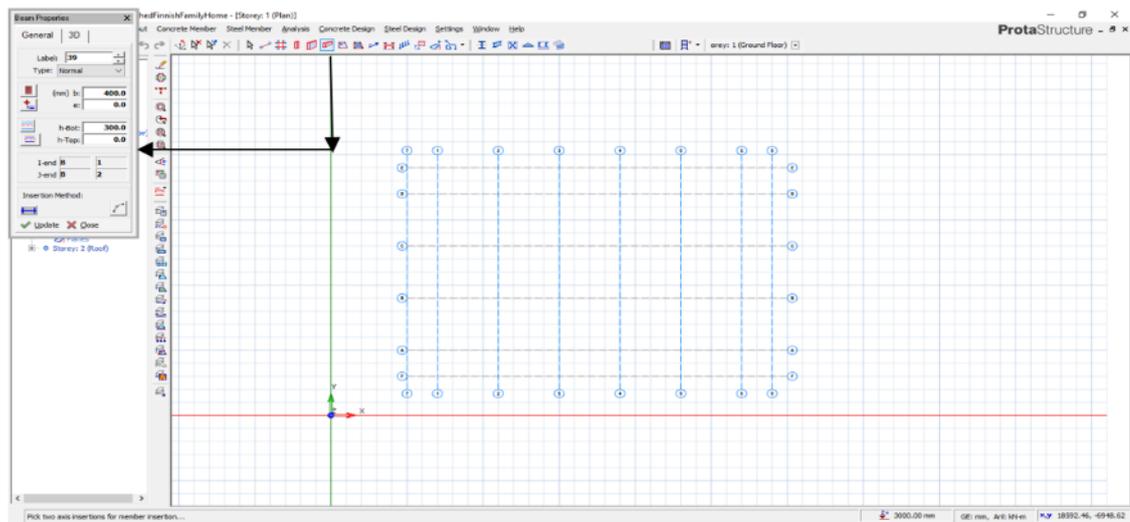

Figure10. Beam Properties with all parameters. [7.]

Other parameters include b as the width of the beam, e as the offset of the beam with the respect to insertion axis, h-Bot as the depth of the beam with respect to the slab top and finally h-Top as the value that allows the adjustment of the top level of the beam. if the value of h-Top is positive the actual beam depth will be h-Top + h-Bot, and vice versa. So, if the value of h-Top is a positive value (100 mm) and h-Bot is 400 mm, then 100+400=500 mm, and if the value of h-Top is a negative value (-100 mm) and h-Bot is 400 mm, then -100 mm+400 mm=300 mm.
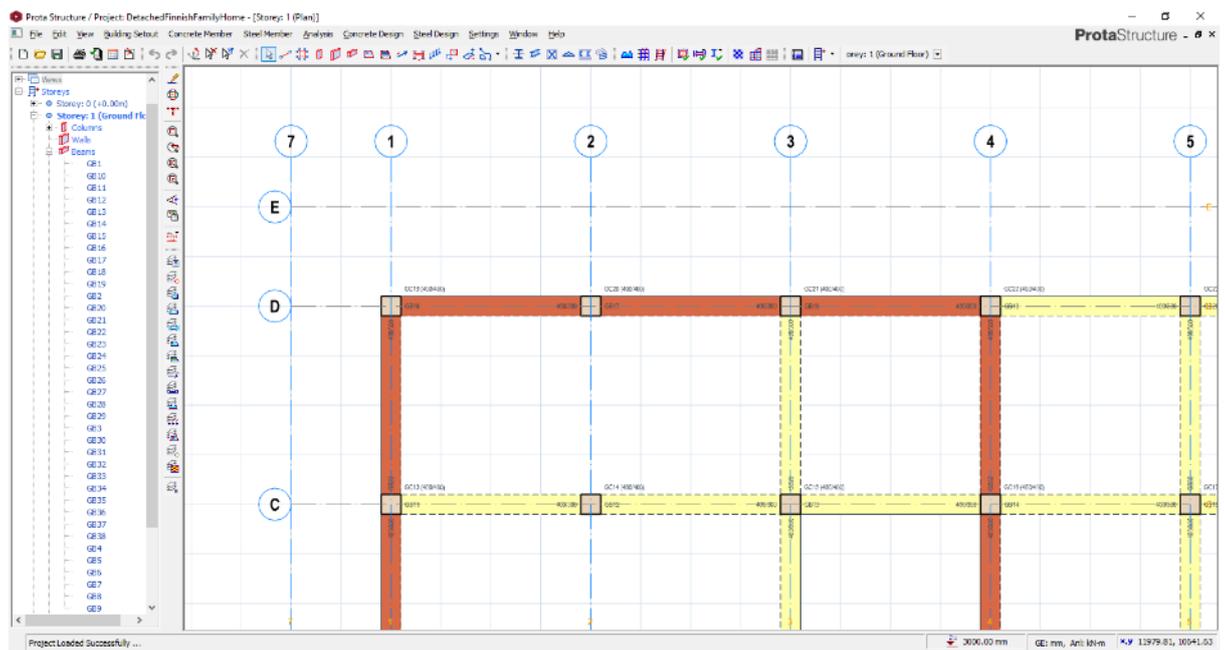
Figure11. First Floor beams on Prota Structure . [7.]

For the single-family detached house in this final year project, the beam dimensions for b and h-Bot were 400 mm and 300 mm respectively while e and h-Top both had a zero value as shown in figure 11. [7, pp.17-20.]

Creation of Slabs

Before creating a slab, it is possible to set up additional loads for it. The icon slab additional load allows the definition of the materials used for the slab area. These loads can then be saved and applied as slab additional loads or service loads. The added loads for the house in this final year project included damp proof membrane, finishing concrete, insulation material and timber floor finish as prescribed in the architectural drawings. This

process was achieved by selecting building setout drop-down menu and selecting slab additional load library. Then, additional loads were added and saved as described in figure 12 which shows slab additional load set up for a flat roof. [7, p.21.]
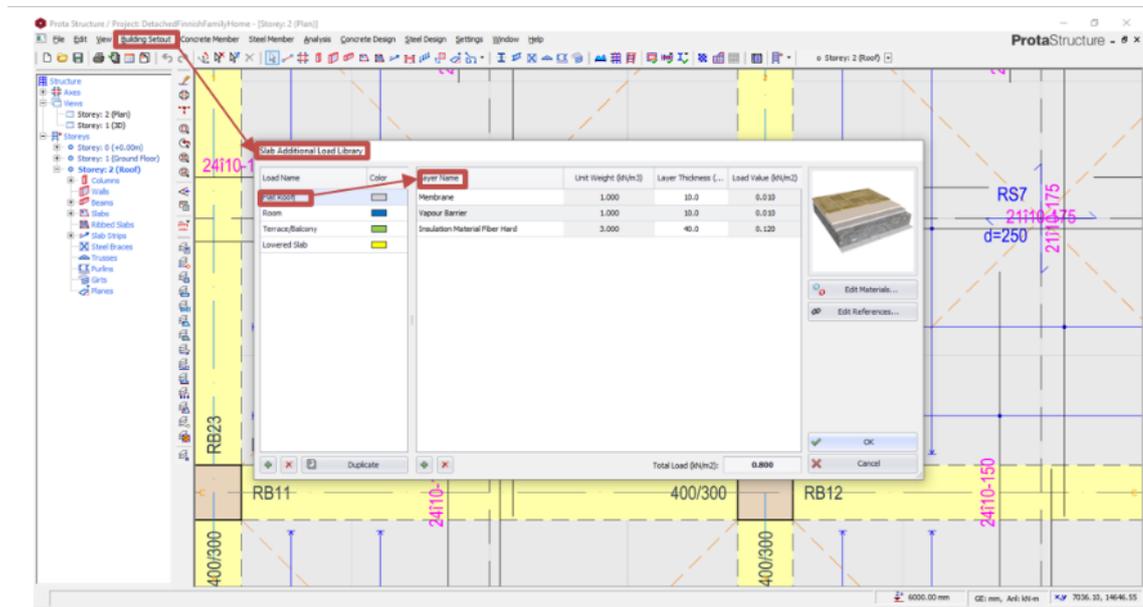

Figure 12. Set up user defined load for a slab additional load. [7**.**]

The intermediate slab thickness for final year project was 200 mm and all materials used for the entire area of the first-floor slab were the same. However, the roof slab (flat roof slab) contained the following additional materials that had to be laid one after another: a 10 mm thick high-performance membrane vapor barrier, a 40 mm thick fiber hard insulation material and finally a 10 mm thick water proof membrane. These additional slab loads can be chosen to design a 250 mm thick flat roof as seen in figure 13.
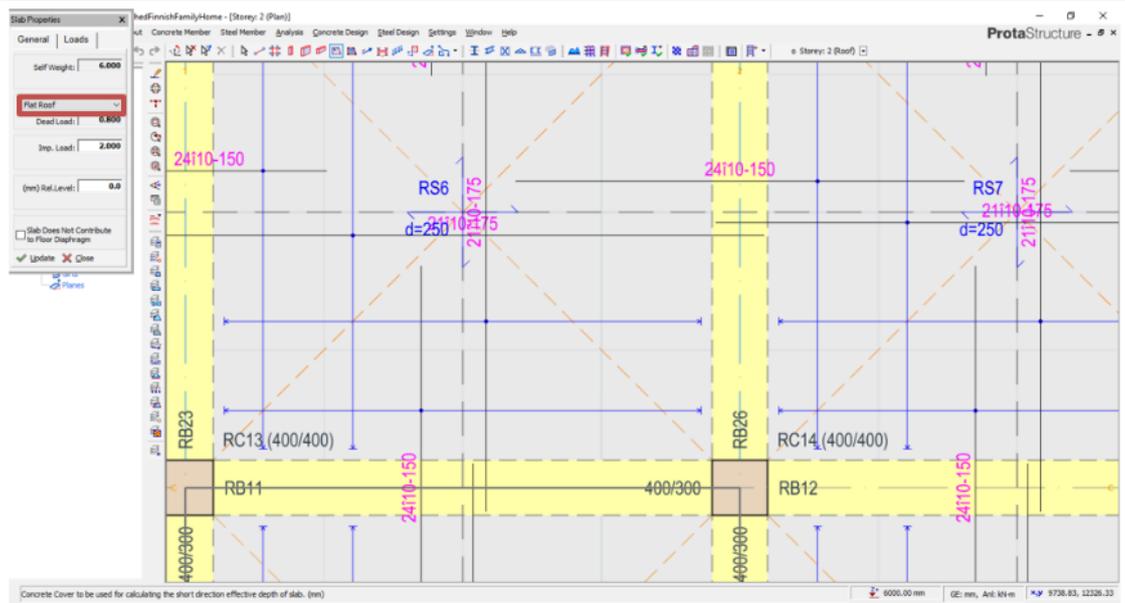
Figure 13. Additional slab menu for flat roof. [7.]

The concrete cover for all slabs in the detached house used in this project was 25 mm, their self-weight was 3.6 KN/m^2 for the first floor slab and 6 KN/m^2 for the flat roof. The dead load which can be either calculated automatically or entered manually and imposed load which can be chosen by right clicking on the imposed loads icon with various dead loads according to space are listed were 2.375 KN/m^2 and 2.00 KN/m^2 respectively, for the first-floor slab. The flat roof had a dead load of 4.560 KN/m^2 and Imposed Load of 2.000 KN/m^2. After entering all load values using the slab properties menu as shown in figure 13, a slab was placed with an insertion icon called beam region. This means that, all the slabs were supported by beams as shown in figure 14. [7, pp.21-26.]
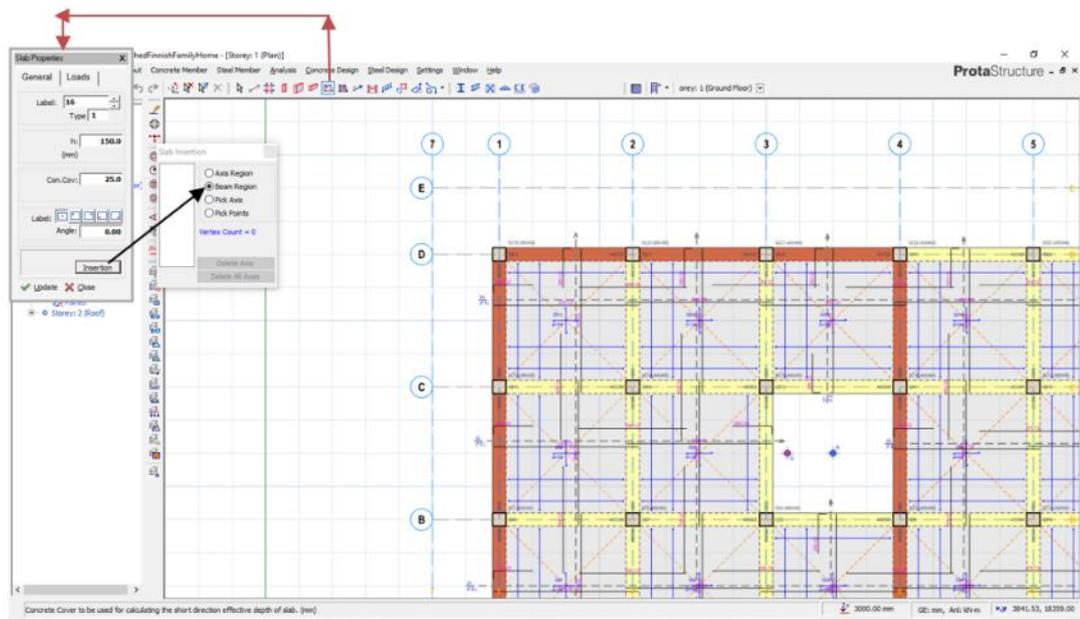
Figure 14. Slab properties with the selection beam region for the slab insertion. [7.]

Beam Wall Loading

The beam wall load was set up by to a building setout drop-down menu, and then the beam wall load library was selected. However, the default wall load materials did not comply with the architectural drawing's details, and so a new beam wall load with 400 mm total thickness, called exterior wall, was created. [7, pp.27.] The exterior wall load (7.999 KN/m^2) had the following layer materials shown in the figure 15.

- Plaster Gypsum 6 mm thick
- Air Space 20 mm thick
- Reinforced Concrete 180 mm thick
- Insulation Mineral (soft) 94 mm thick; and
- Concrete 100 mm thick
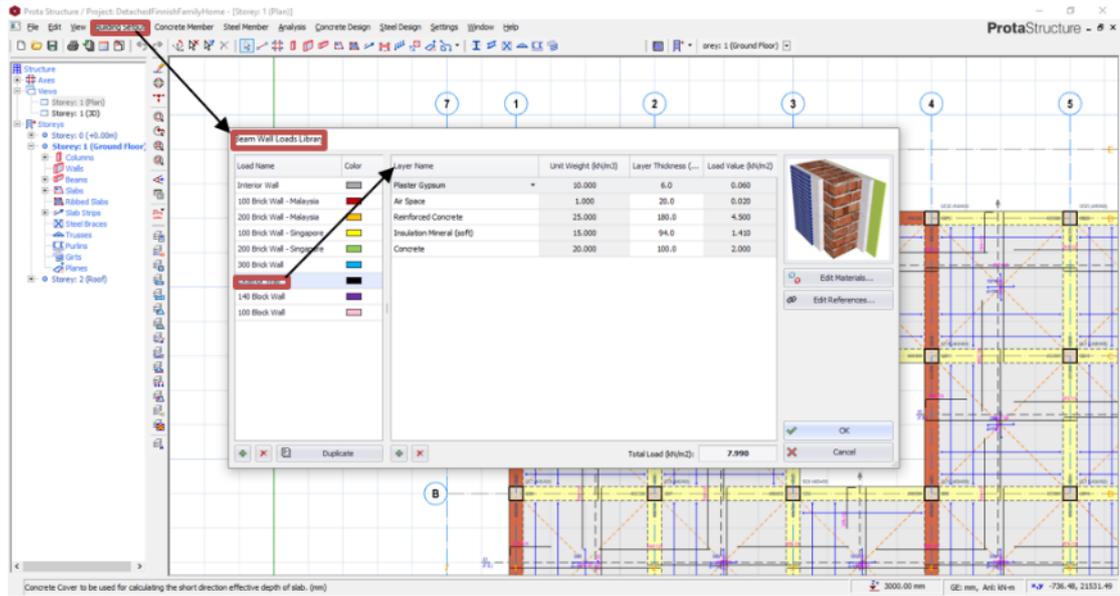
Figure 15. Exterior wall on Beam Wall Load Library. [7.]

After setting up the exterior wall from the beam wall load library for the house used in the thesis, the next task was applying exterior wall load on the beams. This was achieved by right clicking on the particular beam and then selecting edit beam wall load where the load profile editor would appear with the following exterior wall details. [7, pp.28.]

- Wall Unit Weight 7.990 KN/m^2
- Wall Height   3.000 m
- Wall Thickness 0.400 m, and;
- Dead Load G: 23.97 KN/m (this is basically a Uniformly Distributed Load UDL comes from the multiplication of Wall Unit Weight and Wall Height)
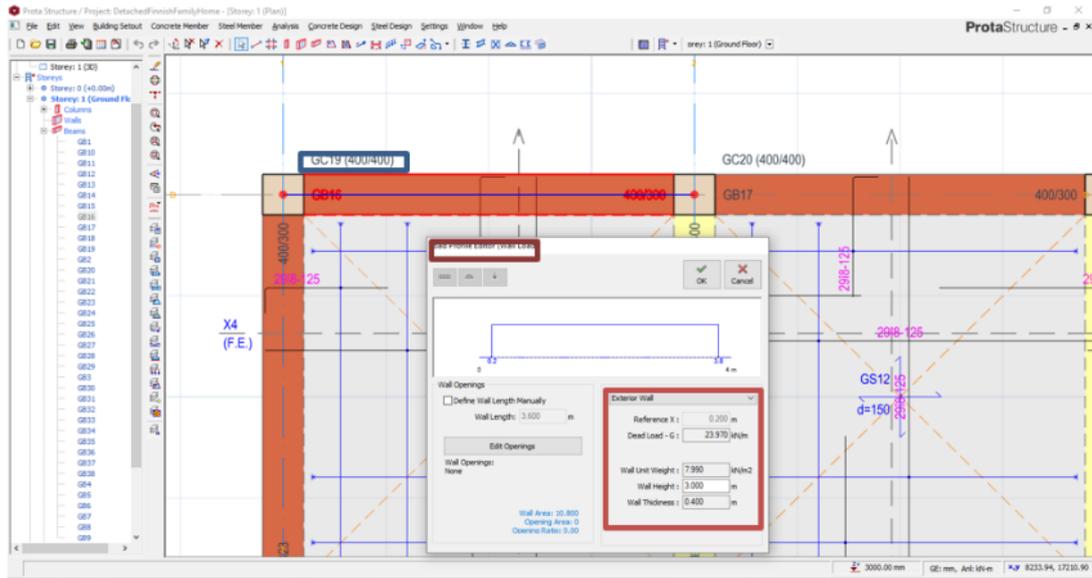
Figure 16. Load profile editor for the wall load for a beam CG19. [7.]

After the load profile editor process was completed, the beam CG19 was re-highlighted with a right click command to copy and paste the beam wall load throughout the entire beam perimeter as shown in figure 17. [7, pp.27-28.]
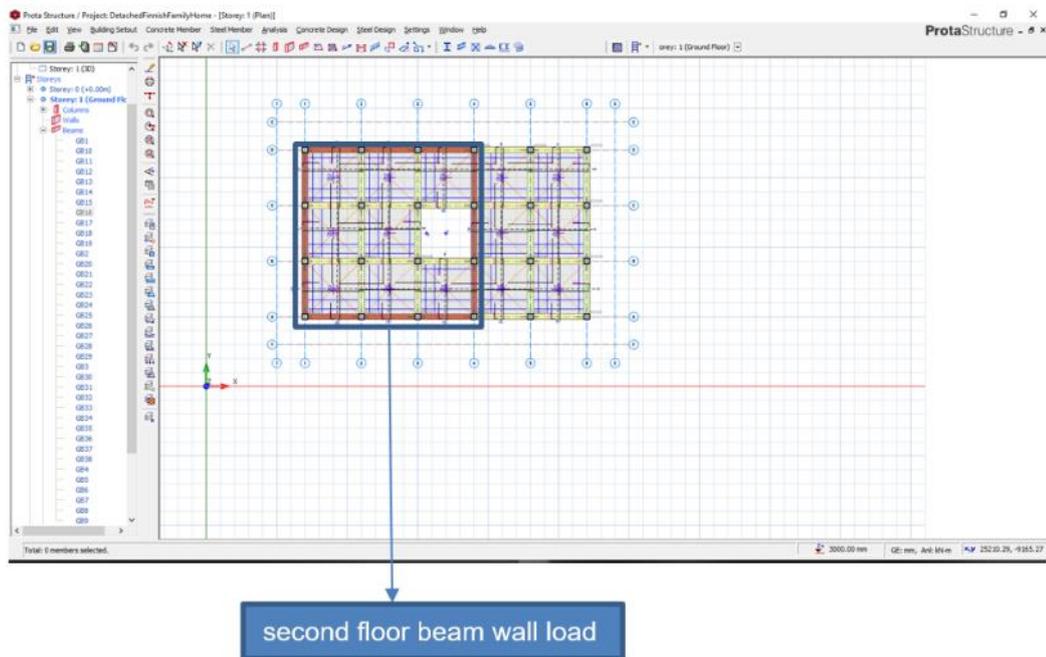

Figure 17. Flat roof (Second Floor) beam wall load applied. [7.]

5.4   Raft Foundation

The design of the raft foundation for the single-family detached house used as an example in this project began by drawing the axes for the area of foundation. This was achieved by firstly, offset all edge axes to 2000 mm outwardly. For instance, in figure 18, from axis D to E, a grid was outwardly extended by 2000 mm and likewise from axis 1 to 7, a grid was outwardly extended by 2000 mm. This was followed by picking up the slab icon and subsequently 480 mm depth of slab (raft foundation) and 50 mm clear concrete cover were chosen, and then by using pick axis insertion method, the raft foundation area was created.

Apart from a 400 mm thick reinforced concrete layer, the raft foundation also had additional slab loads of a 4 mm thick water proof membrane, 74 mm thick unreinforced concrete and 2 mm of timber floor finish. This eventually makes slab total thickness up to 480 mm.
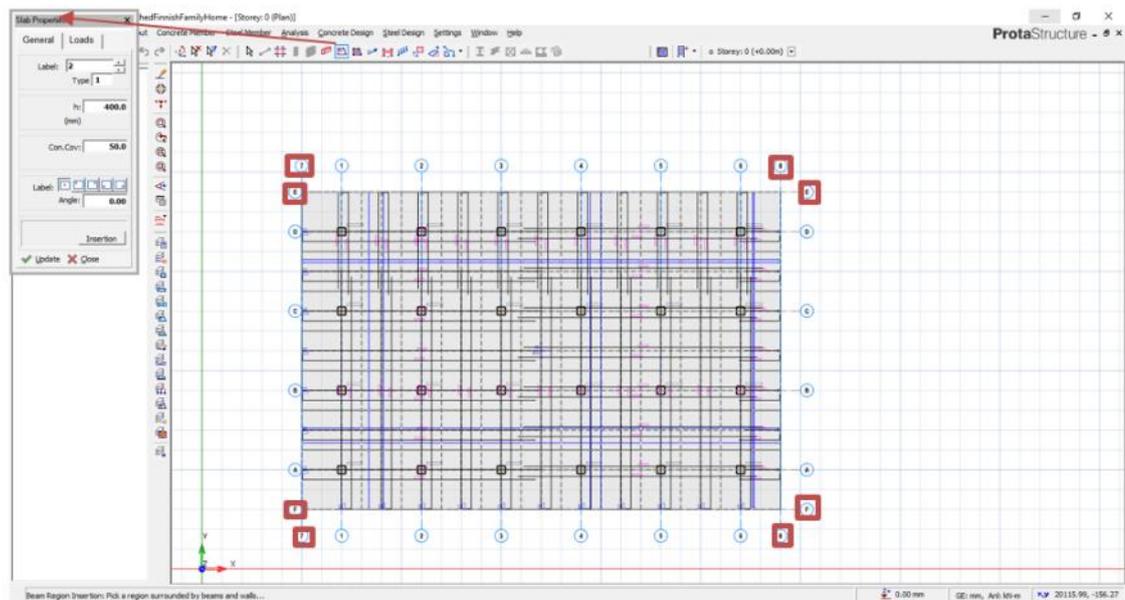

Figure 18. Slab properties with all parameters for a raft/mat foundation. [7.]

The insertion of the raft foundation properties was followed by running a foundation analysis using FE foundation analysis icon just to ensure no errors were existing. However, before running this analysis, it was essential to also choose the right size of reinforcement with edit contour tool. The size of the reinforcement was chosen with the

help of the required steel areas, user defined contour icon and the edit contour tools. Both had a 10 mm diameter for both the top and the bottom reinforcement area.

Figure 19 below shows the As(d)1-bot, (the area of the steel/reinforcement diameter distributed longitudinally at the bottom 1). This area contained 785.40 mm^2, 10 mm diameter of reinforcement and 100 mm center to center. The process of choosing the reinforcement bars was similarly performed in determining the area of steel diameter distributed longitudinally at the bottom 2, which was As(d) 2-bot, as well as for the area of steel diameter distributed transversely at the both top 1 and 2. The top and bottom areas of steel diameter distributed transversely are termed by As(d) 1-top, and As(d) 2-top.
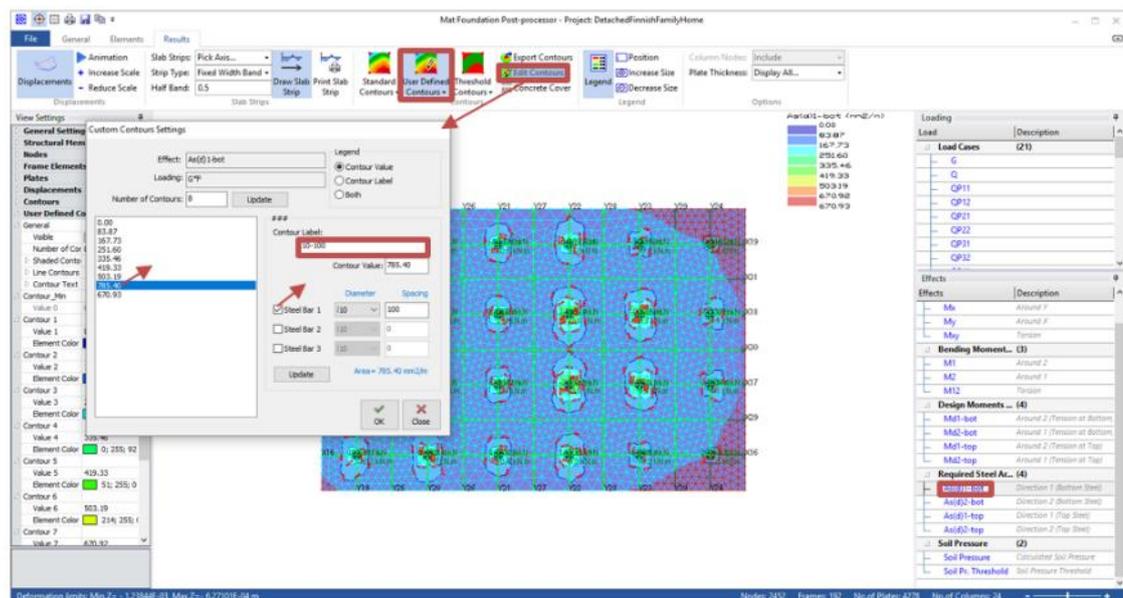

Figure 19. Using contour to choose number of reinforcements required by area of steel. [7.]

The parameters selected for the raft foundation analysis included both column sections in the FE Model icon to check punching shear and consider beam torsional stiffnesses for torsional checking. The analysis was successful, and no errors were found, as indicated in figure 20. Next, the slab strips were drawn both vertically, in the Y-direction and horizontally, in the X-direction. [7, p.43.]
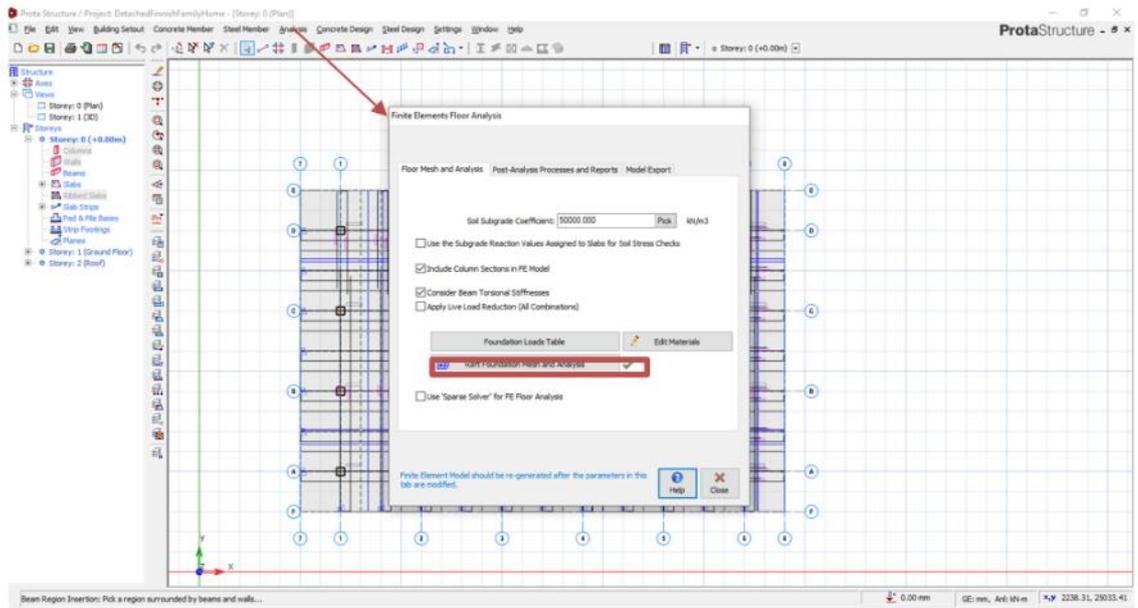
Figure 20. Green tick indicating a successful raft foundation analysis. [7.]

The final year project continued by creating the slab strips in both X and Y direction. This was followed by a re-run of the raft foundation analysis to check if there were any errors. From then on, slab strip FE analysis results were automatically transferred into the model and hence the reinforcements for the raft foundation were created as can be seen in figure 21 below. [7, pp.42-44.]
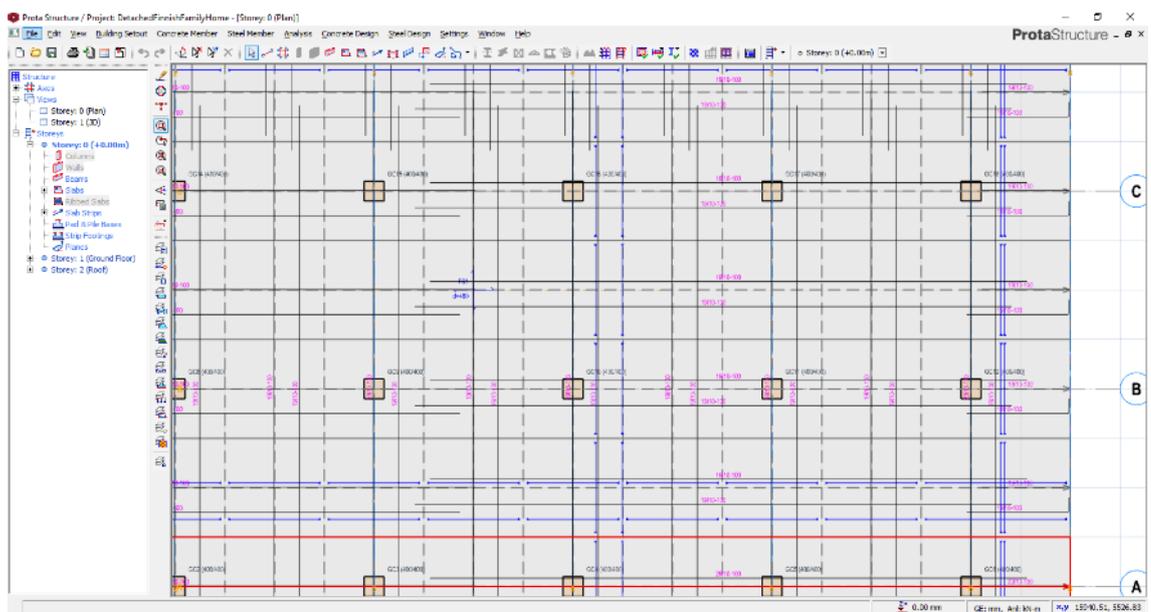

Figure 21. Raft foundation reinforcement layout. [7.]

6   Building Analysis

After creating all the essential members of the sub structure and super structure, the next step in the final year project was to analyze the model and perform checks to detect errors. The task started by displaying the Prota Structure building analysis user interface as shown in figure 22. The pre-set parameters were set up with the pre-analysis icon. The main assumptions of a Building Analysis are;

- Parameters to review and modify the design codes and notional horizontal load, foundation soil allowed stress, lateral loading and lateral drift.
- Load Combination to add or modify load combination sets.
- Storey Loads and Parameters to review or add inputs to any later loads such as wind load.
- Edit Materials to review or change the concrete grades and reinforcement material. [7, p.29-30.]
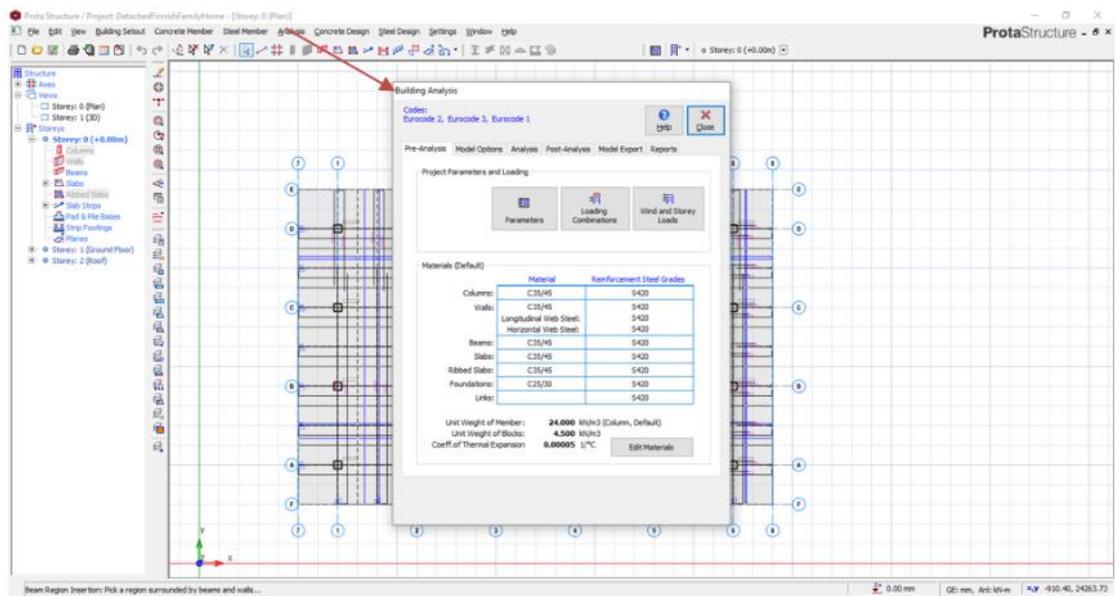

Figure 22. Set up parameters for running Building Analysis. [7.]

The codes used for designing in the final year project were;

- Eurocode 2 EU (EN 1992 for Design of Concrete Structures),
- Eurocode 3 EU (Design of Steel Structure) and;
- Eurocode 1 UK (EN 1991 Actions on Structures)

 The concrete grade used for the detached house in the thesis was C35/45 for all members including columns, walls, beams and slabs. However, the concrete grade for

foundation was C25/30 loading combination tool, was used to auto-generate the load cases and load combination as shown in figure 23. [7, pp.30-33.]
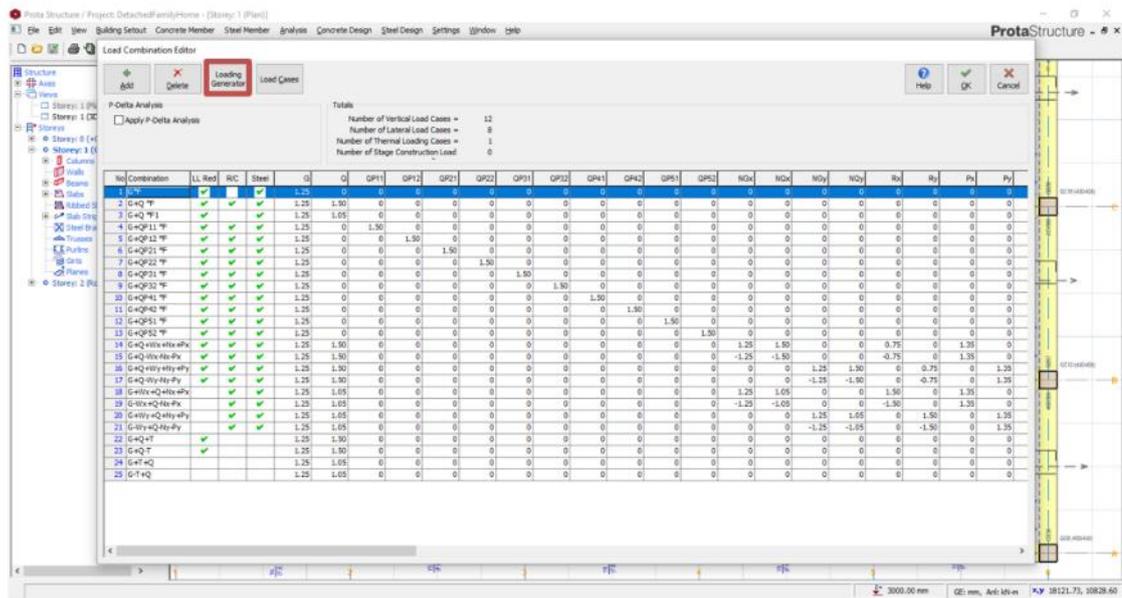
Figure 23. Load combination generator as is shown on a model. [7.]

6.1   Running Analysis

A building model must be checked for its validity before running an analysis. In the final year project, this process was achieved by using the building model check button in the Prota Structure. The building model check helped to determine if there were any overlapping slabs, columns, slab and beams, beams and walls or any columns inside slab panels, as well as the validity of axis intersections, proximity of axis intersections and overlapping columns spanning more than one storey. If any of these overlapping conditions occur during the process, returning to the graphic editor and modify the members is inevitable. However, this was not the case for the final year project as there were no detected overlapping conditions. [7, p.33.]

After running the building model check, the building analysis was successful performed. This was done by also selecting column/wall reinforcement design and beam reinforcement design as a part of running building analysis. In the time of running the building analysis, the beam loads were automatically calculated and accomplished, slab

loads distributed onto supported beams, the weight and the mass of each storey calculated and any notional horizontal loads determined. The features of running building analysis are highlighted in figure 24 below.
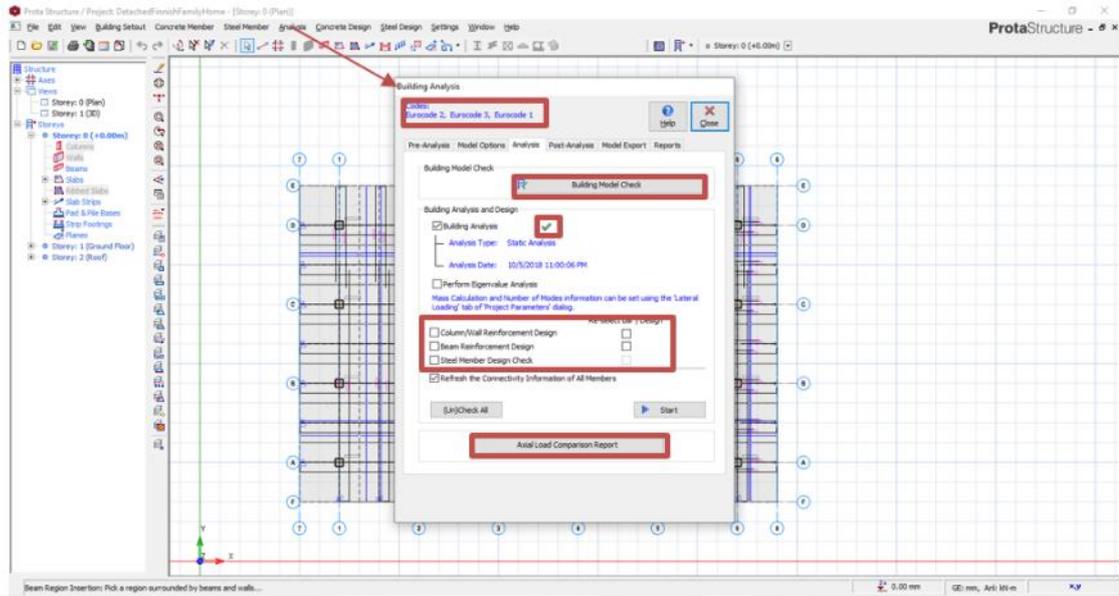
Figure 24. Building analysis successful completed. [7.]

The analysis results for the final year project are discussed in detail in chapter 6.2 and 6.3. The very first step after running a successful building analysis was to check the axial load comparison report. The main purpose of the axial load comparison report was is to see the summation of all dead and live loads utilized at each floor and to see the axial forces displayed numerically into both pillars and walls. The loads were cross checked with a Load In=Load Out check. If these values do not agree one another at a tolerance value of 5%, an investigation of any missing loads and must be done, and missing loads must be added. Then a new   building analysis must be done, and the report needs to be checked again. [7. pp.33-34**.] The phenomenon of the axial load comparison report is usually accompanied with three tables. Tables 4a, 4b and 4c below, and the definition of each table is explained in the text below.

   Table 4a, represents the weight of the model. It is also the sum of dead and live loads of all members when the slab is yet decomposed on the beam. Table 4b is a summation of decomposed or calculated on slab loads to beams, other applied loads on beams, and beam self-weights. The values under the slab column are shown as zero values, because the slab loads have been automatically calculated to the beams by yield line or

finite element decomposition method. In addition, table 4b is vital for the building analysis as it tells whether all slab loads have been successfully captured into the beams. [7, p.34.]

Table 4c is basically a summation of all axial loads in the columns and walls derived at each storey level after building analysis. The forces are the actual output forces, and so the table shows whether these axial loads have been successfully captured. [7, p.34.]

The values prorated in tables 4a, 4b and 4c should agree so that, for example, the sum of table 4a should be equal to a table 4b and the sum of table 4c should also be equal to 4b.

Table 4a had total dead loads and live loads values of 5203.3 KN and 928.0 KN, respectively. These numerical values were equivalent to the summation of dead loads and live loads values of the table 4b. This meant the table 4a and 4b agreed to one another and there was no slab lost.

   Table 4a total dead loads (5203.8 KN) =Table 4b total dead loads (5203.8 KN), and
   Table 4a total live loads (928.0 KN) =Table 4b total live loads (928.0 KN), passed.

 If it happens table 4a is greater or equal to a table 4b. This means that the loads are conservative and so it is optional to recheck the model but not necessary.

Table 4c had a total column and axial loads (delta G) of 5203.8 KN and (delta Q) of 928.0 KN. Also, these numerical values were equivalent to those total values of table 4b.

    Table 4b total dead loads (5203.8KN) = Table 4c (delta G) total column and wall axial loads (5203.8 KN), passed.
    Table 4b total live loads (928.0 KN) = Table 4c (delta Q) total column and wall axial loads (928.0 KN), passed.
 If table 4c total values are greater or equal to a table 4b total values. This means that, loads are conservative. The loads can be rechecked, but it is not necessarily important. These facts can also be substantiated in table 4a-c below of Axial Load Comparison Report [7, p.34.]

Table 4a-c. Axial Load Comparison Report [7,p.34.]

a total load based on slab loads

G-dead loads

Storey	Column	Wall	Beam	Slab	Ribbed Slab	Total
1	276.5	0.0	110.1	1708.3	0.0	2094.9
2	276.5	0.0	1225.2	1607.2	0.0	3108.9
Total						5203.8

Q-live loads

Storey	Column	Wall	Beam	Slab	Ribbed Slab	Total
1	0.0	0.0	0.0	480.0	0.0	480.0
2	0.0	0.0	0.0	448.0	0.0	448.0
Total						928.0

b total loads decomposed to beams

G-dead loads

Storey	Column	Wall	Beam	Slab	Ribbed Slab	Total
1	FE mesh					2094.9
2	276.5	0.0	2832.4	0.0	0.0	3108.9
Total						5203.8

Q-live loads

Storey	Column	Wall	Beam	Slab	Ribbed Slab	Total
1	FE mesh					480.0
2	0.0	0.0	448.0	0.0	0.0	448.0
Total						928.0

c Building analysis column and wall axial loads

Storey	G	Delta G	Q	Delta Q
1	2094.9	2094.9	480.0	480.0
2	5203.8	3108.9	928.0	448.0
Total		5203.8		928.0

6.2   Post-Analysis Results

One of the features of the building analysis tab user interface that requires to be examined is a post-analysis tab, the post-analysis tab allows all model results to be reviewed in a graphical manner, and in fact, this is where the model and analysis results display can be checked. By default, the deflected shape is shown in red and the un-deflected shape is blue-colored. Also, load cases and combinations could be chosen using different icons available on the right-hand pane. [7, p.35.]

Using the results tab and switch off a displacement tab, would allow to only see the analysis model. The model will be shown in blue-colored center lines as shown in figure 25. The first thing to check when analyzing the detached house used in the final year project was the members connectivity. This was done by scrolling down the mouse wheel or holding down with mouse pan and zooming in to a location of interest. [7, p.35.]

It is also possible to turn on and off the information about nodes and elements in the elements tab. For example, turning on node labels displays the node numbers, and turning on loads, supports, springs and diaphragms displays the chosen information. In addition, the frame elements` icons would allow to display more information of beams, columns and walls. These inputs were vital in examining the members connectivity, and, in deed they helped whether a check of the physical model translated correctly and accurately into an analytical model. [7, pp.35-36.]
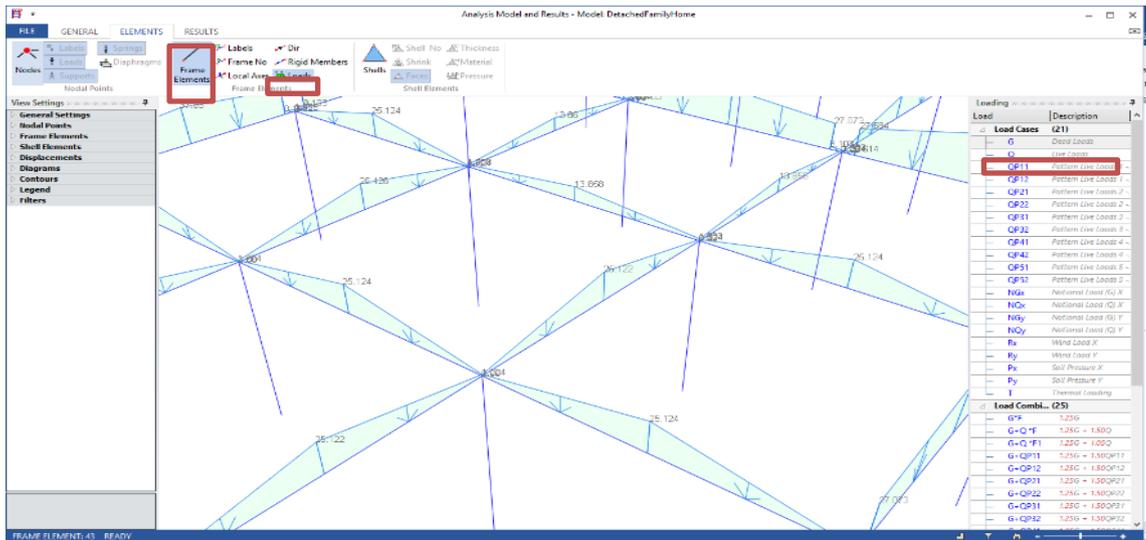
Figure 25. Model analysis results, in blue. [7.]

Once the connectivity of the members in the detached house used in this final year project was successfully confirmed and done, the next step was to check the loadings on the model in detail. For example, by choosing a load case G and selecting the load icon on the frame elements tab, all loading affecting the slab that are calculated onto the beams will be displayed; this check was done to ensure that all the beams have slab loads in them and are supporting slabs. This could also be proven by checking if the values are reasonable, by looking at the symmetry of the analytical model, and performing simple hand calculations. This task was performed to all load cases and load combinations of the model. [7, p.36**.**]
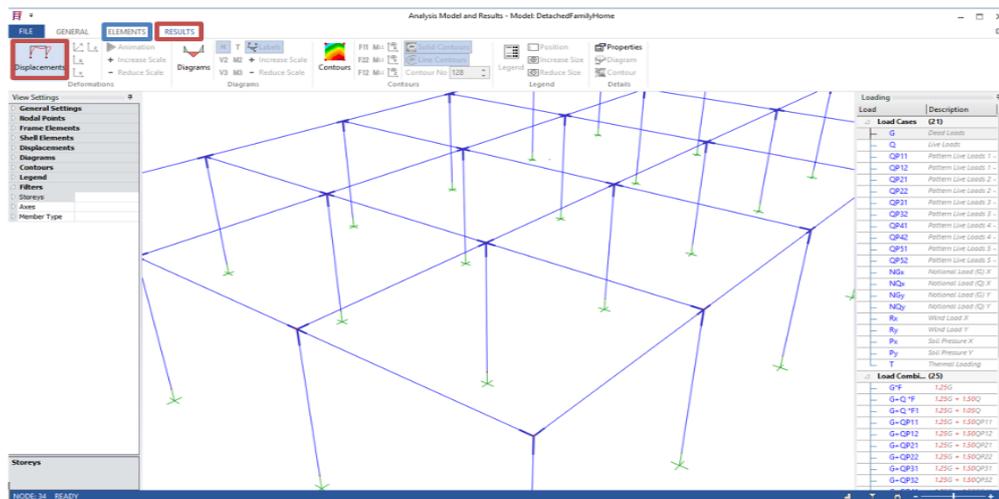
Figure 26 Showing Analysis Model Results in blue color wide frame. [7.]

6.3   Model displacement

It is essential to check model displacement when checking the connectivity and the resultants of the members. The task was performed by going to the results tab in the Prota Structure and then selecting displacement tab as indicated in figure 27. The displacement will be shown in red and that was where animation tab and resultant tab were selected. The use of animation tool would allow a better visualization of the displacement. This was very useful when checking the connectivity of the members: if a member was disconnected, it would be displayed separately. [7, pp.36.]

It was also important to check the exact magnitude of resultant. Using the resultant tab displays the absolute resultant of a joint in 3D space: this is the resultant of displacement in X, Y and Z direction. It was vital to check if the values of displacement are reasonable and expected. In all accounts, the displacement reflects the forces that a member experience. Therefore, if the displacement is unreasonable, the forces on the members would also be doubtful. Figure 27 below shows the successful displacement values after selection a load case G which shows the maximum and minimum displacement for the model was 0 mm and 1 mm, respectively. [7, pp.36.]
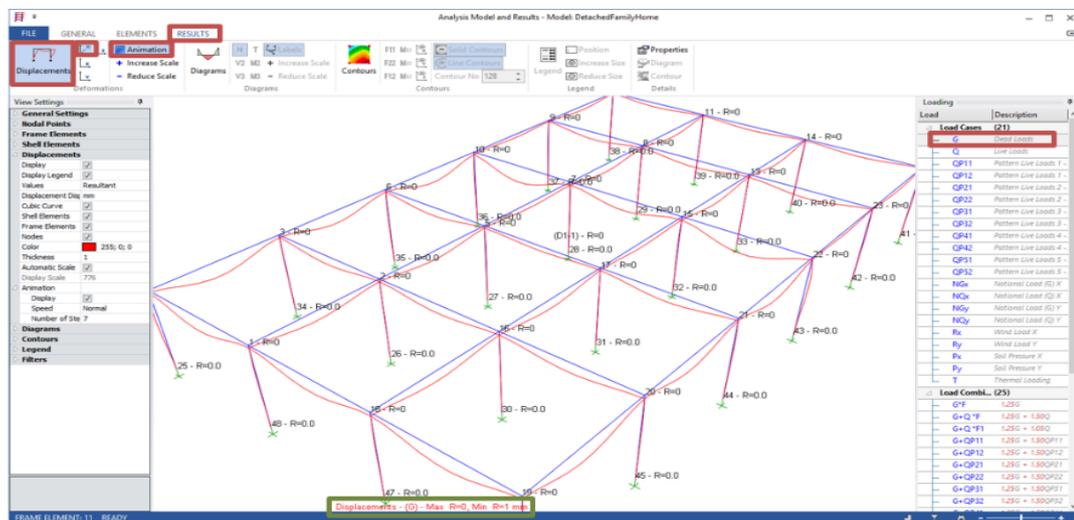

Figure 27. Displacement values for a load case G. [7.]

Finally, a check for displaying the actual member forces was done in the analysis. Using the display diagrams icon available on the model analysis and result display in the Prota Structure, the various forces and moments were analyzed. These forces and moments

included N, V2, V3, T, M2 and M3. Table 5 below shows the detailed descriptions of all forces and moments involved. [7, pp.36**.]**

Table 5. Forces and moments [7.]

Forces	Moments	Description
N	not applicable	Axial forces
V2	not applicable	It is a major shear force of the beam, it is also a horizontal shear of the column on the plan view. In addition, it is also a major shear of the wall along the stronger direction.
V3	not applicable	It is a minor shear of the beam, it is also a vertical shear of the column along on the plan view. Moreover, V3 is also a minor shear of the wall along the weaker direction.
	M2	It is a minor moment of the beam, it is also a moment of the column in direction 2. In addition, M2 is also a minor moment of the wall along the weaker direction.
	M3	It is a major moment of the beam, it is also a major moment of the column in direction 2. In addition, M3 is a major moment of the wall along the stronger direction.
T	not applicable	Torsion forces

For example, on N= Axial Forces, by selecting G dead load cases, as can clearly be seen in figure 28, the G axial load increases as it moves towards the Raft Foundation. Hence, the results are as expected. Similarly, other results were displayed for V2, V3, T, M2 and M3 for a G dead load as well as other 20 load cases and 25 load combinations.
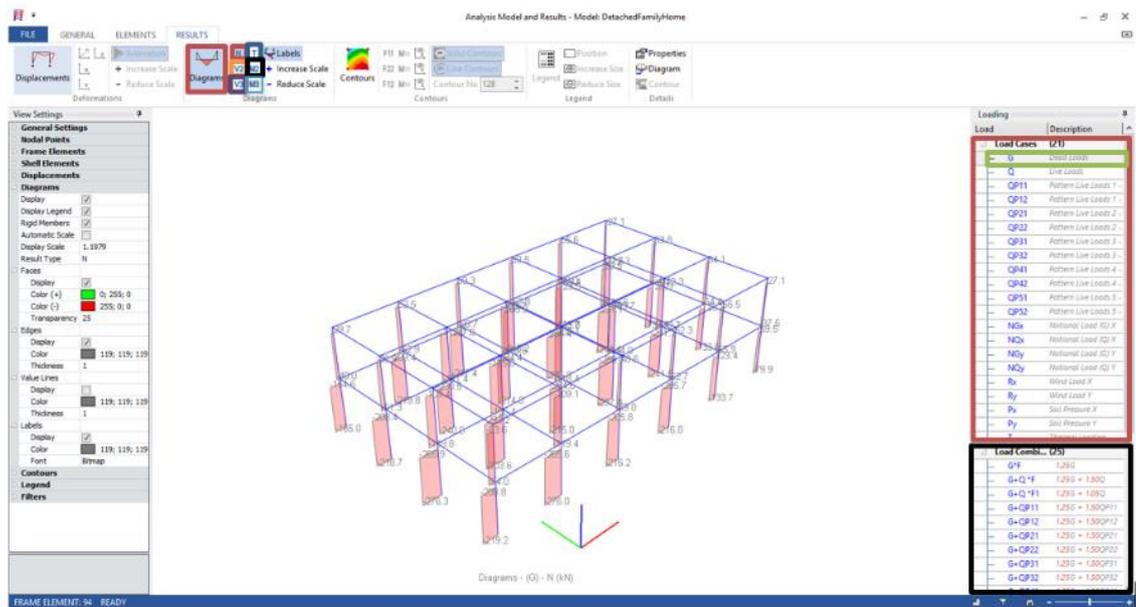
Figure 28. displaying diagram icon for N (axial forces) by choosing G load case. [7.]

All 21 load cases and 25 load combinations for this final year project were analyzed and checked. Before proceeding with any task of designing members, it is essential to examine the analysis model and results display closely. This should be done along with other checks described in table 6 below.   [7, pp.35-37.]

Table 6. Checks to be performed during model analysis [7.]

Checks to be performed	Descriptions
Analytical wire-frame	First, check if all members are connected properly, and secondly check whether the analysis model created accurately based on what has been modelled.
Loadings on the model for each load case	First, check if all slab loads are calculated correctly onto the beams, and secondly check whether all lateral loads applied correctly.
Displacement of the model for each load case	Since, the displacement reflects the forces members can experience. Therefore, any unreasonable displacement would indicate model forces are erroneous.
Members forces for each load case	First, check by verifying axial, shear, moment are reasonable and if there is any doubt. Turn on the displacement icon, as it says the displacement is a reflection in which forces members can experience.

6.4   Column and Beam Design

Running reinforcement design of the columns, wall and beams were selected as a part of Building Analysis. All the columns and beams were passed according to the designing codes that were listed in the beginning of chapter 6. Therefore, using a post-analysis tab, which also displays column design and beam design icons. The reinforcements for columns and beams were designed. The number of bars for each column were eight and each bar was 20mm in diameter, plus 8mm diameter bars for links. For example, by using an interactive design tool, a column labeled as GC1 (storey 1) was examined in more detail. As can be seen in figure 29, section dimension reveals different column`s details, e.g. sizes, height and concrete cover, steel bars reveal the sum total of rebar and its layout, load combination table gives the list of all load combinations, required and supplied area of reinforcement (As) are computed from the maximum load combination, and slenderness tab reveals the bracing presumption and computed distance between the inflection points in the column when it bends.   [7, p.37.]
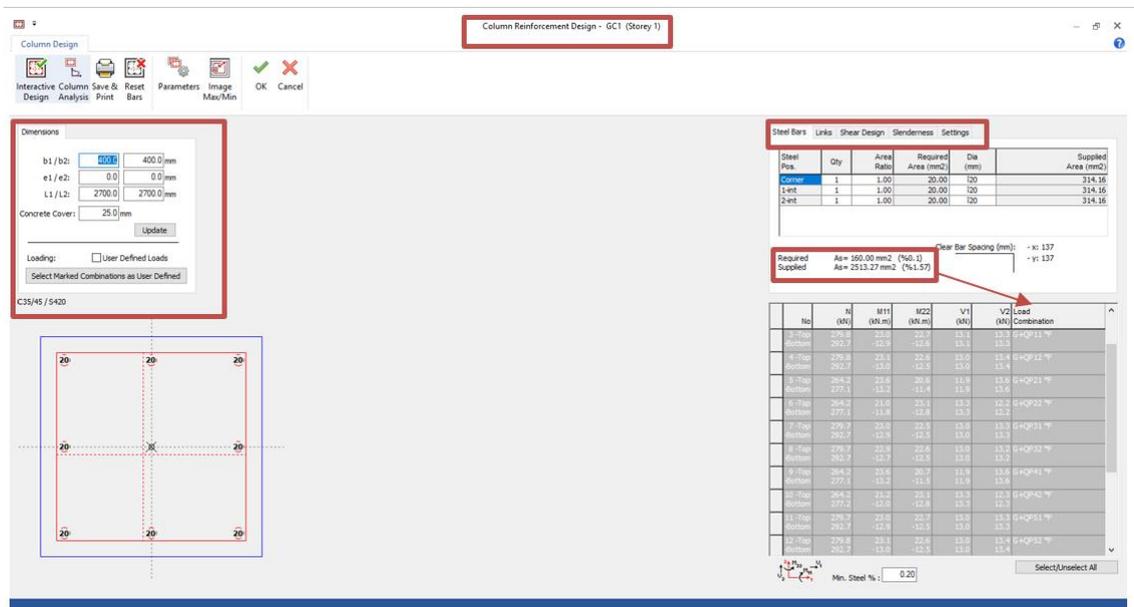
Figure 29. All columns designed and passed successful. [7.]

Also, the beams for the detached house used in this project were designed using the same method. The size of the reinforcement for the beam was 20mm in diameter and the size for links was 8mm in diameter, with 175 mm center to center. Figure 30 gives the reinforcement data for a beam axis A as indicated below: the required area for reinforcement As is outlined in blue since it is based on design envelope of all load combinations, extra As discloses the supplementary or excess area of steel based on the chosen type and size of reinforcement steel, rebars designed are disclosed in the different rows as well as shear reinforcements (these can be manually modified if required), s-Bar (top and bottom) discloses the clear cover distance of the rebars at the uttermost layer, and finally Deflection Check uncovers the actual vs allowable span/effective depth. [7, pp.37-41.]
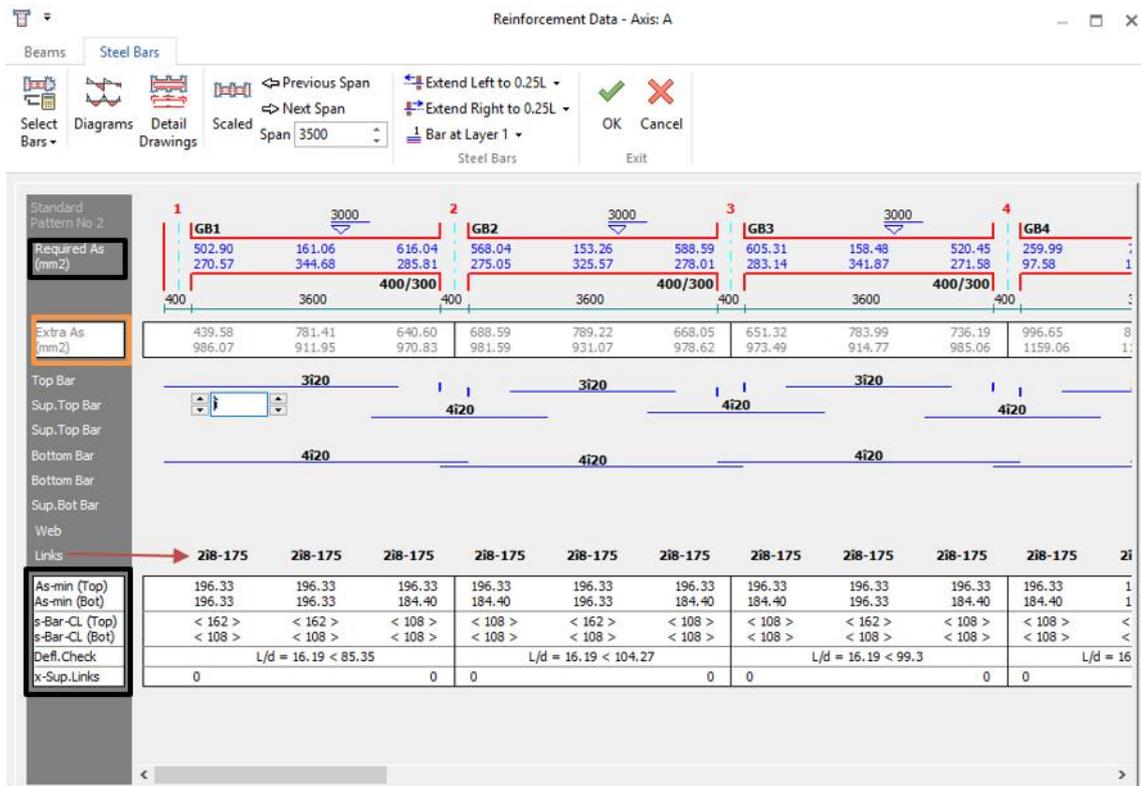
Figure 30. Showing all beam designed and passed successful. [7.]

6.5   Slab Analysis & Design

For slab models, slab reinforcements are designed after the insertions of slab strips both in X and Y directions. This process is achieved by using the moment of coefficient method from established tables. This was an independent process, and therefore, was carried out after the general building analysis. [7, p.44.]

Furthermore, all slab types were set correctly in accordance with the established tables. This was achieved automatically in the batch mode of the slab icon in a structure tree and eventually selecting the altenative Set Slab Types Automatically. Using slab strip type called FE Strip (design based on the Code), the slab strips were inserted in X and Y directions. Then the Slab Analysis and Design, followed by picking a Design icon as can be seen in figure 31 were choosen on the concrete design menu. [7, pp.44-42.]
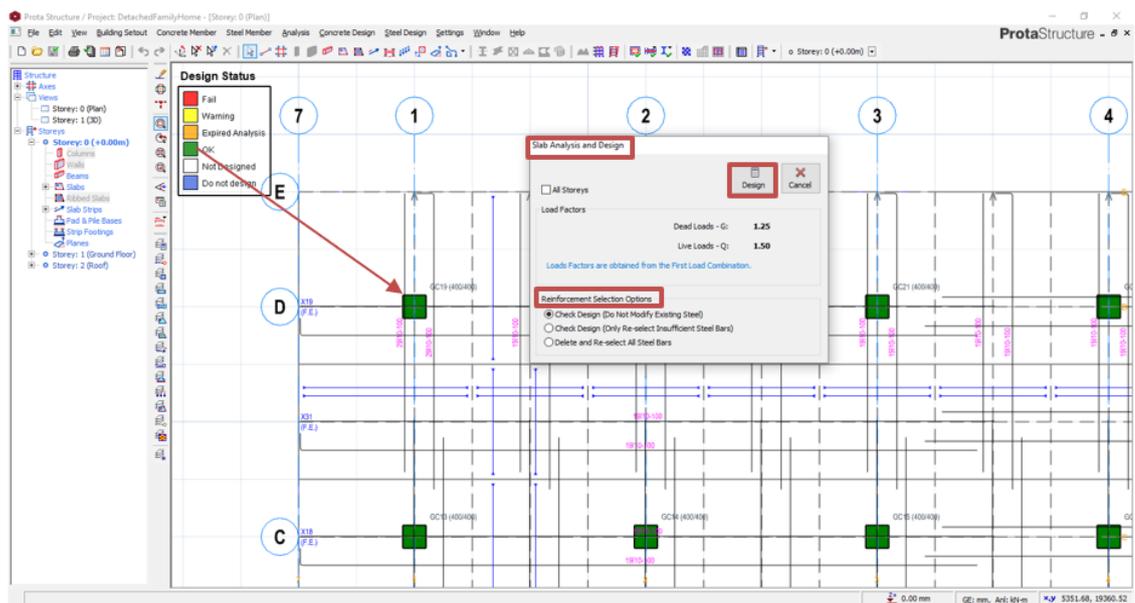

Figure 31. Slab analysis and reinforcement design with, design status green for OK. [7.]

Figure 31 shows the slab analysis and reinforcement design, with design status green for OK. [7.] The results disclose a reinforcement size of 10 mm diameter is distributed both longitudinally and transversely at 100 mm center to center

distances, for a bottom area of the steel/reinforcements, as well as a top area of the steel/reinforcements. [7.]

7   Prota BIM Integration

When a structural modelling task in Prota Structure is completed, the following task is to share the structural model into Revit. This is an essential step for a BIM integration between the two applications.

Before explaining the term Prota BIM, it is valuable to list the methods of BIM integration. The term `integrated, in construction industry could be defined as a comprehensive approach to a building where all key people of a project work in a profoundly collaborative way throughout the project`s facility life cycle to achieve the intended results towards the buildings. In addition, the lists for methods of BIM integration are outrighted in table 7 below. [9.]

Table 7. The lists for methods of BIM integration [9, p.6.]

Methods of BIM integration	Description
Custom Link (bi-directional link)	**High level**; good enough to perform the task it was designed for (Prota Structure→Autodesk Revit
Open BIM (IFC)	**Good level**; detailed information of product with its geometry and physical characteristics (all industries)
CSI/2	**Medium level**; mainly object data (CIMsteel, AISC)
SNNF	**Medium level**; object data (only for steel industry)
ACIS (AutoCAD)	**Low level**; extruding different shapes.
DWG, DXF	**Low level**; no object intelligence

Prota Structure and Autodesk Revit use mainly custom links called bi-directional links to enhance project coordination and workflow. Thus, these two applications must be in place before even starting the modeling structural design task.

The structural engineer uses Revit and the DXF (drawing exchange format) file produced by an architect. Therefore, with the help of Revit application and the DXF file, the structural engineer will start to model the structural design by with Prota Structure. Once the structural design is done, the model is imported into Autodesk Revit. This aspect will enable changes to be made before the model is exported back to Prota Structure. Therefore, using Autodesk Revit to export and import of structural model allows strengthening the coordination between disciplines and parties associated with a project. [9.]

Table 8. Shows the advantages of using Prota BIM for bi-directional link. [8, p.36.]

Prota BIM bi-directional links benefit(s)	Description(s)
Importing/Exporting model	Prota structure model can be exported and imported into Autodesk Revit. The process is fast and easy, and a model can be easily accessed in Revit; with no intermediary file.
Multi-material supports	It supports multi-material for example the design concrete and steel buildings
Non-limitation	The bi-directional link has no limitation in geometry, it can even support sloping slabs
Flexibility	The bi-directional link is flexible when it comes to what is needed to be imported.
Changes in Revit model	All changes in Autodesk Revit model can be transferred to Prota Structure, and the Prota structure will automatically update the change.
Sharing with other parties	Using Autodesk Revit, the Prota Structure model can be shared to other parties such as Contractor, Electrical and, Client, Architect, etc.

Prota Structure uses a Prota BIM ribbon available on Revit to integrate the model interchangeably. This enables changes to be synchronized and followed closely in Autodesk Revit as well as Prota Structure. The advantages of using Prota BIM for bi-directional link are listed on table 8.

8   Conclusion

As the main theme of this final year project was to design structural concrete members, the tasks of modelling reinforced concrete structural members for a detached family home were successfully executed using Prota Structure. In addition, all beams, columns, slabs and the raft foundation were analyzed to detect if there were any errors.   All analysis checks including an axial load comparison report, analytical model, axial forces, major shear, minor shear, major moment, minor moment and torsion, were passed using designing eurocodes.

Firstly, the checks for a member's connectivity were successfully performed onto a location of interest such as column and beam joints, zooming the analytical model and rotating it into a different direction to view the connectivity more closely. The key was to examine whether the physical model that had been set up translated correctly and accurately into an analytical model.

Secondly, the results of load cases and load combinations from the model were justified by verifying that all beams had slab load values in them. The validity of these values was furtherly examined by looking at the symmetry or by simple hand calculations.

Thirdly, animating the displacement, and displaying the resultant tab was crucial for analyzing the analytical model. This helped to better visualize the model, and had any member disconnected within the model, it would have displayed separately. Furthermore, checking the exact magnitude of the displacement was fundamentally important, using the resultant tab of absolute displacement of a joint in 3D space. The resultant tab contains the values from the X, Y and Z components. The values showed a maximum number of 1mm which was reasonable.

The final check was done to verify the members for each load cases. This check was successful because all moments and forces involved in the model were reasonable. The

study also explained briefly how the Prota Structure can exchange information with Autodesk Revit. The phenomenon is called bi-directional link, and, in many ways, it can significantly enhance project coordination and workflow. It is crystal clear; a bi-directional link with Revit is advantageous for project coordination and synchronization between structural engineers, project architects, owners and other stakeholders.

Finally, the outcomes of this thesis can be used for further study of Prota Structure. More research has to be conducted, however, regarding Prota Structure. Therefore, this thesis could be used as one of the references.

References

1   De Ros Reyes, Adrian. 2006. The role of computer-aided drafting, analysis, and design software in structural engineering practice. Online. Massachusetts Institute of Technology. < http://hdl.handle.net/1721.1/35080>
    Accessed 7 June 2018.

2   Integrated Analysis and Design Solution for Building Analysis. Prota Structure. Online.
    2018.<http://protasoftware.com/documents/protastructure_2018_brochure.pdf.>
    Accessed 3 May 2018.

3   Heidenry, Margaret. 2018. What is Single Family Home. Online. Realtor.
    <https://www.realtor.com/advice/buy/what-is-a-single-family-home/>. Accessed
    12 October 2018.

4   Working with DSA Architectes. DSA Architects. Online.
    <http://www.dsaarch.com/design-process/> Accessed 1 November 2018.

5   Understand Building Construction. 2018. Raft or Mat Foundations. Online.
    <http://www.understandconstruction.com/raft-foundations.html>. Accessed
    3 June 2018.

6   User Guide 2018. Composite Structures Dialog Box. Online. Graphisoft.
    <https://helpcenter.graphisoft.com/user-guide/65858/> Accessed 20 June 2018.

7   Quick Start Guide. 2018. Online. Prota Structure.
    <https://www.scribd.com/document/369073977/ProtaStructure-QSG-En>
    Accessed 10 April 2018.

8   What`s new. Enhanced Revit Integration. 2018. Online. Prota Software.
    <http://www.protasoftware.com/documents/protastructure_2018_whats_n
    ew.pdf> Accessed 9 September 2018.

9   Kubin, Joseph. 2015. Prota Structure. BIM. Online.
    <http://www.thefcic.org/files/files/565c3f69b833d.pdf> Accessed 12
    September 2018.

10 Henttinen, Tomi. 2012. Common BIM Requirement series 3. Online. Gravicon
    Oy.<https://buildingsmart.fi/wp-
    content/uploads/2016/11/cobim_3_architectural_design_v1.pdf>.
    Accessed 20 November 2018.

detached single family house	STUDENT VERSION (0000000)
Axial Load Comparison Report 1 26.05.2018 Rev: 1	Calc. By: DM Checked By: DM

Axial Load Comparison Report

TOTAL LOADS (Based On Slabs Loads):

G - Dead Loads:

Storey	Column	Wall	Beam	Slab	Ribbed Slab	Total
2	276.5	0.0	110.1	1708.3	0.0	2094.9
1	276.5	0.0	1225.2	1607.2	0.0	3108.9
Total						**5203.8**

Q - Live Loads:

Storey	Column	Wall	Beam	Slab	Ribbed Slab	Total
2	0.0	0.0	0.0	480.0	0.0	480.0
1	0.0	0.0	0.0	448.0	0.0	448.0
Total						**928.0**

TOTAL LOADS (Decomposed to Beams):

G - Dead Loads:

Storey	Column	Wall	Beam	Slab	Ribbed Slab	Total
2	FE Mesh					2094.9
1	276.5	0.0	2832.4	0.0	0.0	3108.9
Total						**5203.8**

Q - Live Loads:

Storey	Column	Wall	Beam	Slab	Ribbed Slab	Total
2	FE Mesh					480.0
1	0.0	0.0	448.0	0.0	0.0	448.0
Total						**928.0**

BUILDING ANALYSIS COLUMN AND WALL AXIAL LOADS:

Storey	G	Delta G	Q	Delta Q
2	2094.9	2094.9	480.0	480.0
1	5203.8	3108.9	928.0	448.0
Total		**5203.8**		**928.0**

Total Base Reactions: G = 5203.8 kN   Q = 928 kN