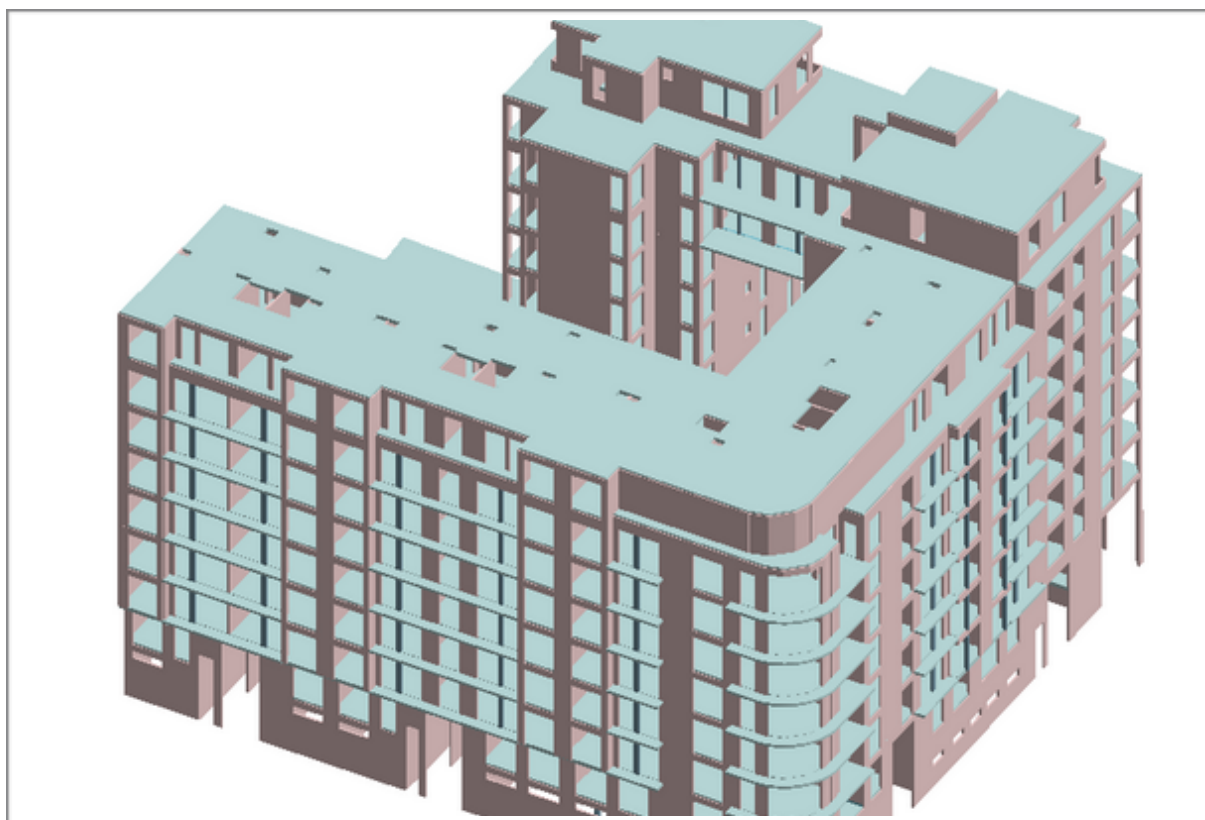


Aalborg University

Master Thesis

Design collaboration between Finite Element Analysis and Architectural model



by

Zuzana Mikurčíková

© Aalborg University, Spring 2015

Zuzana Mikurcikova

The contents of this report is freely accessible, however publication
(with source references) is only allowed upon agreement with the author.

This report is typeset in Baskerville 11pt

Layout and typography by the author using Mac Pages.

AALBORG UNIVERSITY
STUDENT REPORT

Master Thesis, Master of Science
at the School of Civil Engineering
Fredrik Bajers Vej 5, 9100 Aalborg
Telephone [99 40 99 40](tel:99409940)
Fax [99 40 99 40](tel:99409940)
<http://www.ses.aau.dk/>

Title: Design collaboration between Finite Element Analysis and Architectural model

Project period: February -June 2015

Supervisor: Kjeld Svidt

Number of pages: 45

Turned in: 10 June 2015

Participants: Zuzana Mikurcikova

Synopsis:

The aim of this master project of the M.Sc. program in Structural and Civil Engineering is to analyse the possible automatisisation of the information transfer between architectural design and finite element analysis. Firstly theory behind finite element analysis is drawn out and processes of the FEM software used in this report are described. Then a sample building is created in Autodesk Revit, handling the design of the structure. The export of the model from Revit is launched and the data are transferred into FEM-design software, which handles the finite element analysis of structural model. This data transport is possible through StruSoft add-in, that enables the information transfer of analytical model data. Finite element analysis is conducted on the imported model resulting in changes in properties of the load-bearing system. The changes done on the model in FEM-design need to be loaded back into Revit where the original model is updated. The processes of data transfer between the two softwares are analysed and the limitations are discussed. The possible challenges that the process can face are identified.

Preface

This master thesis has been written as part of M.Sc. program in Structural and Civil Engineering at School of Engineering and Science at Aalborg University by Zuzana Mikurcikova.

Special thanks goes to thesis supervisor Kjeld Svidt, Associate Professor at the department of Civil Engineering, for guidance and supervision throughout the project. Gratitude also goes to Iwona Budny, Development Engineer at StruSoft DK and Dennis Kristensen, Vice President of StruSoft DK for providing information and learning material on the subject's matter.

The thesis was carried out during the 4th semester in the period of February to June 2015 with the title *Design collaboration between Finite Element Analysis and Architectural model*.

Table Of Contents

Preface	4
1. Introduction	6
2. Project Information	8
2.1. Problem Statement	8
2.2. Project Scope	9
3. Theory behind FEM-design	9
3.1. Analysis Calculations	9
3.1.1. Finite Element Calculations	11
3.1.2. Finite Elements	13
3.1.3. Finite Element Mesh	20
3.1.4. Load Group Combinations	24
3.2. Design	24
4. Testing	24
4.1. Creating Model	24
4.2 Applying Loads	27
4.3 Design and Analysis	28
4.4 Finite Element Mesh	30
4.5 Results	31
5. Revit to FEM-design Connection	32
5.1 Basic Principles of Import and Export	33
5.2 Workflow	34
6. Conclusion	42
7. Bibliography	45

1. Introduction

“An efficient construction industry is critical to our growth and prosperity in the future.”

Minister for Climate, Energy and Building, Rasmus Helveg Petersen

Danish government is implying on creating stronger and growing construction industry, creating more job opportunities and better conditions for firms. Growth in construction industry and dealing with international competition push construction companies to use better organised working environment. This engage improving collaboration between involved parties, time management, creating more accurate models, reducing construction costs and assuring safety at construction site. The development and use of information technology is helping to do better job at these tasks. [kebmin.dk, 2014]

Nowadays firms purchase various CAD software to deal with multi-disciplinary designs, the practices are compartmentalised and the communication is still represented in 2D drawings. This can be time consuming, labour-intensive and error-prone. To create better interpolation between disciplines architectural, engineering, construction and facility management can be combined in building information model (BIM). The early definition that BIM is a 3D model of a facility is not legit anymore and as defined by the NBIMS (National BIM standards) document "A BIM is a digital representation of physical and functional characteristics of a facility. As such it serves as a shared knowledge resource for information about a facility forming a reliable basis for decisions during its lifecycle from inception onward." Therefore any information that need to be shared about a building can be made available digitally and stored in a cloud where stakeholders might access them at any time. These can include owners, designers, engineers, contractors, bankers, estimators, etc. All these people could access the important collaborating data needed for completing, maintenance and demolition of a structure [nationalbimstandard.org, 2015].

From all the compartments meeting the most critical is collaboration between architectural design and structural analysis. The traditional process uses 2D CAD drawings of architectural design to classical structural finite element model (FEM). This transition is time consuming, job-demanding and can create design errors, all this will lead to increase in costs. The architectural model differs from structural FEM in many ways, therefore data transfer has to be precise and well understood. The interpolation between them can be seen in Figure 1.1. As the figure indicates the architectural model is composed of non-structural and structural member information. Structural member data are shared with structural model seen as overlapping circles. The structural member information consists only of the members designed to carry loads, all other members are referred to as non-structural. Now the structural model usually FEM contains, in addition to load bearing member system, information about loading cases, their combinations the structure might have to sustain during its lifetime, as well as data about the geometric boundary conditions, etc. The load bearing member system that connects the two models are special building elements such as beam, column, slab or wall. These models have to correlate and the common members shall be identical. With the BIM solution this can be achieved [Ling, Xue-yuan, Xi-la, 2011].

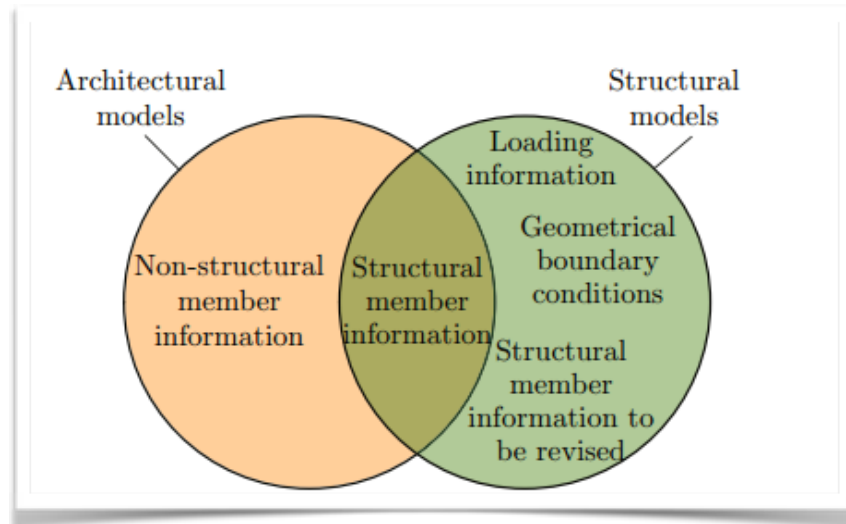


Figure 1.1. Relationship between data of the architectural and structural models [Ling, Xue-yuan, Xi-la, 2011]

This report will focus on the interpolation between the structural FEM and architectural model using BIM technology and software available on the market today. A structural model of a sample building will be created. Then the model data will be transferred from Autodesk Revit, a building design software specifically build for BIM, to FEM-design a finite element analysis software made by Strusoft. This is possible through StruSoft Add-In that transfers the analytical model of a structure, the process is visualised on Figure 1.2.



Figure 1.2. Link between Autodesk REVIT and StruSoft FEM-Design [www.strusoft.com, 2010]

2. Project Information

2.1. Problem Statement

Information technology is being more often implemented into practice in construction department, collaboration of architectural design a structural modelling is crucial for successful project delivery. Having different disciplines handling specific parts of the project may lead to errors in the design of a structure. This project focuses on the finite element model, analysing the structure's load bearing components, and its connection with the architectural design of the building. This is affecting the work flow and cooperation among structural engineers and architects. Architects using design tools creating a model in Revit, where a member like a wall is made with certain geometry, covering, material properties, colour etc. The same wall has to be taken by structural engineer and analysed in finite element program for load-bearing capacities as a wall made of certain type of concrete with suitable reinforcement. Then the wall is tested by load cases and combinations that can occur. Depending on results of the analysis a structural engineer may have to change geometry or material of the wall. Again this update on design should go back to the architect and be changed in his/hers model. This process is repeated until the final design is reached. Classical method of this process consists of manual information transfer by printing out 2D drawings and comparing them at a meeting. Nowadays this process is being automatised. Different solutions are available on the market right now, as this automatisation of data transfer is new, challenges and issues can occur along the way. This project is focused on the potential benefits and obstacles that the process leads to. The following questions will be answered throughout the report:

How does the use of model updating handled by BIM help to the structural engineers and architects?

- How far is the process automatised and what remains manual work in the information transfer?
- What are the limitations of integrating a model from Revit to FEM-design and vice versa?
- Is this modern method easily implemented into the ongoing workflow?

2.2. Project Scope

This report will look into one of the possibilities on how to improve the data exchange. StruSoft a danish company introduced a Revit add-in that makes possible to export and import an analytical model of structure's design. The structural software involved is FEM-design, which handles finite element analysis and design of steel, concrete and timber elements. Firstly a peak of theory behind the FEM-design is going to be presented. Describing all the possible calculations the software is offering to the structural engineer then the finite elements, finite element mesh and load combinations are introduced and described. Later the functionality of the software is tested. A sample model will be created in FEM-design, following video tutorials and manual available. Later finite element mesh will be applied and refined, the load cases and combinations will be created. After this the design of structural members is carried out, the sections and materials of steel columns is designed and then the concrete elements are design with suitable reinforcement. The analysis will be made and the results displayed. This is done in order to understand the design procedures. When having the models ready the information transfer and the procedure will be described. Then the actual data exchange will be tested. First a model from Revit using the StruSoft add-in export tool will be used to transfer the entire analytical model into FEM-design. Then the data transfer from FEM-design back to Revit will be tried out using the import tool. Finally the StruSoft update tool will be used to proceed back into FEM-design with made changes. At the end the program's limitations and compatibility with Revit software will be summarised.

3. Theory behind FEM-design

In this chapter the applied theory and design behind the finite element modelling of FEM-design software will be briefly described. The focus on processes in FEM-design are based on the manual given by StruSoft [strusoft.com,2010]. This chapter is done in order to better understand processes the software makes and to understand the limitations. The FEM-design can perform finite element analysis calculation and design calculation for load-bearing concrete, steel and timber structures according to Eurocode.

3.1. Analysis Calculations

The finite element analysis (FEA) is an approach used to find a solutions to complex problems for example in structural analysis. This approach does not estimate a real physical problem, which would be far too complicated. Instead of finding an exact solution by solving numerous differential equations, it approximates the structure into finite element model and finds the approximate solution as can be seen on Figure 3.1.1. [Cook et al., 2001].

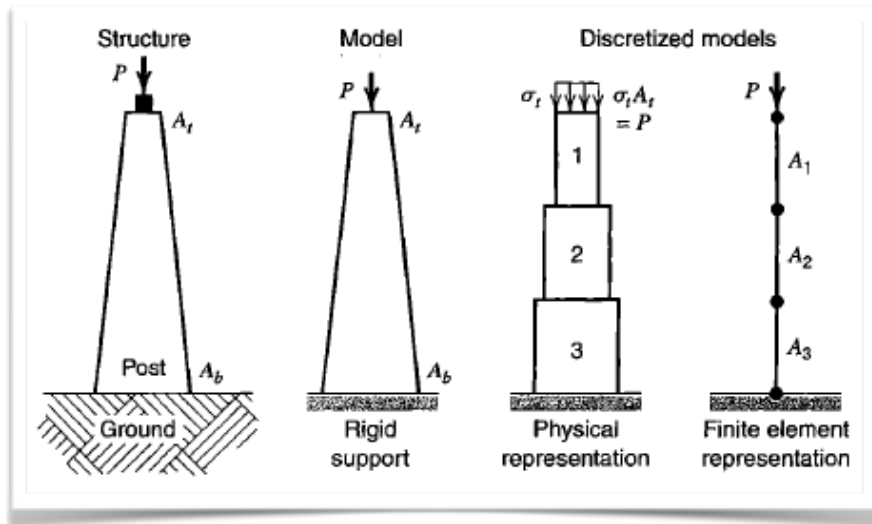


Figure 3.1.1. Converting real physical problem into solvable approximation [Cook et al., 2001]

To do this there is the need of realising the mechanical properties of the solid object, boundary conditions and the forces applied on the structure in order to get the necessary outputs as stress, strains, displacements, etc. The method divides the entire object into parts, finite elements, these are connected by nodes. The arrangement of finite elements is called finite element mesh.[Cook et al., 2001]. Even though the FEA is not exact, the solution can be improved by increasing the number of finite elements. FEA has no geometric restrictions and can be widely used to many field problems. Boundary conditions are not restricted, any portion of body may be supported while any may be loaded. Material properties are not bound by isotropy and can change from element to another, approximation may be improved by finer mesh as can be seen on Figure 3.2.2. [<http://www.colorado.edu>].

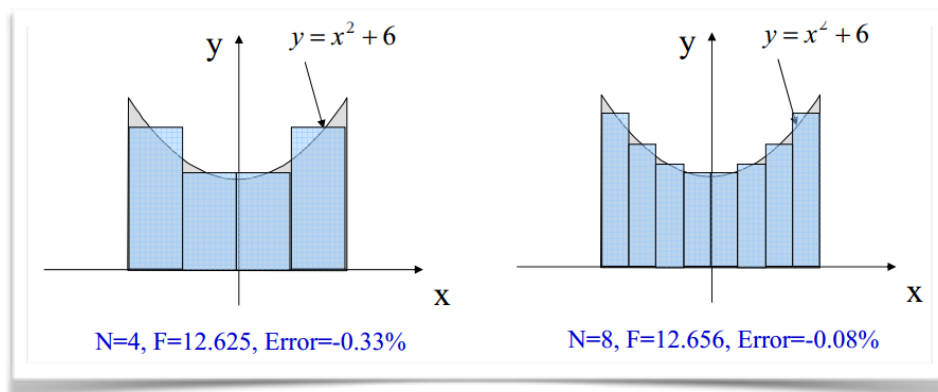


Figure 3.2.2. Minimising error by increasing number of elements [<http://www.colorado.edu>,2006]

FEM-design can perform the following calculations:

- linear static analysis for all structure types
- static analysis with the use of 2nd order theory and stability analysis for spatial structure
- dynamic analysis (vibration shapes, eigen frequencies) for all structure types

- seismic calculation
- non-linear static analysis for supports resisting only compression
- crack analysis- tracking the cracking process

All these calculations are shortly described to see, what the program is able to process, then types of finite elements are introduced. After finite element mesh and the load combinations are studied.

3.1.1. Finite Element Calculations

This section presents all the possible calculations a FEM-design can perform, these start from the basics as simple static analysis and can go to calculations as complicated as seismic analysis. All of them are shortly introduced and described in the following lines.

Static analysis

The linear static analysis gives the node displacements, reaction forces and internal forces or stress of elastic elements. In nodes where several elements of the same kind meet the mean value of internal forces is generated. This is done to avoid discontinuity made by the model, which does not occur in reality.

The linear static analysis is the solution of the below equation system.

$$K \cdot u = Q$$

Where:

K - is stiffness matrix

u - is displacement matrix

Q - is loading matrix, derived from different load cases

2nd order analysis

Linear theory uses the equilibrium conditions which were established before the structure was loaded. When the deformation begins to be larger more accurate solution would be to account with the changes of the structure's geometry. In case of elastic structure the solution to this problem is the second order theory, which is a good approximation for the real state. This theory takes into account only deformations caused by normal forces in relation with bending moment. The stiffness matrix for example of a bar is linearly dependent on normal forces. In case of plane plate:

$$(K + K_G(N)) u = Q$$

Where:

N - is distribution of normal forces

K_G - is geometrical stiffness matrix

The process starts with calculation of normal forces with use of initial stiffness matrix, K , then K_G is found with the found forces, N . At last the modified displacements are found with $(K+K_G)$ matrix. If the load is larger than critical load of the structure the equation cannot be solved.

Stability analysis

The structure's resultant stiffness depends on normal forces, as mentioned above and therefore on loading applied. The structure collapses when these forces lower the stiffness to zero. This becomes an eigenvalue problem.

$$\det(K + \lambda K_G(N)) = 0$$

Where:

λ - is eigenvalue

The structure loses its stability when critical load is acting, this is the product of load and the eigenvalue parameter λ . The smallest value of λ defines the critical load parameter. FEM-design calculates stability by the Lanczos method, resulting in buckling shapes and the corresponding λ -parameter.

Linear Dynamic calculation

In case when acting loads are time dependant, displacements in nodes will also be changing accordingly. Therefore the loads acting on the structure are affected by the structure's momentum, this is defined by the d'Alambert theorem:

$$K u = Q - M u''$$

Where:

M - is mass matrix of the structure

u'' - is the acceleration in a node

Note that no damping is present.

When the loading stops the structure will freely vibrate, the mode displacements are then equal to the following equation:

$$u = A \sin(\omega t)$$

Where:

ω - is angular frequency

A - is vibration shape

Therefore if loading, Q , is zero the original equation of motion becomes, an eigenvalue problem.

$$(K - \omega^2 M)A = 0$$

In FEM-design solving requires iteration process both in 2D and in 3D.

Seismic analysis

FEM-design can also procreate seismic calculation is special type of forced vibration, that is caused by the ground acceleration. Acceleration in such case changes in time and is mostly random. This will cause the structure to vibrate and create inertia forces, which are also time dependant. If internal stresses get too high the structure will fail. Synergy between structure and ground is difficult and assessed by several factors:

- ground acceleration (magnitude, time dependency, direction)
- structures elasticity
- mass of structure and its distribution
- soil and structure association

Non-linear static analysis

- Uplift calculation

FEM-design allows to set resistance only to compression for a point, line and surface supports. This is possible for wall and plate members. Such a case describes material non-linearity. In FEM-design it is firstly tested if tension appears in any supports while loading structure. Then these particular supports assigned as only compression bearing, will have stiffness set for tension to a very small number. Process is done to point when there are no more of these supports under tension.

- Crack analysis in FEM-design Plate

When a specimen is cracked its load bearing state can be also analysed with help of FEM-design with the use of crack analysis technique. This is a non-linear calculation, where superposition does not apply and therefore load groups are not used, but all load combinations must be checked. The analysis is working as iteration where structure is loaded by steps and in each step the stiffness of the specimen is updated as more cracks occur. This changing stiffness applies only in the direction perpendicular to the crack. Parallely the stiffness is unchanged.

3.1.2. Finite Elements

Finite element calculation are carried out after modelling a structural object. To do this the finite elements are used. Choosing the right elements is essential, designer needs to think about how accurate the results must be and also how much computational time and power a problem is worth. The elements that FEM-design offers are represented below. Each one is suitable for something different and therefore need to be understood [www.fem-infos.com,2008].

3D beam

Beam is a structural element that has two nodes. In each node there are six degrees of freedom, three displacements and three rotations, which means that beam elements can take and transmit axial forces, carry transverse forces and bending moments as can be seen in Figure 3.1.2.1.

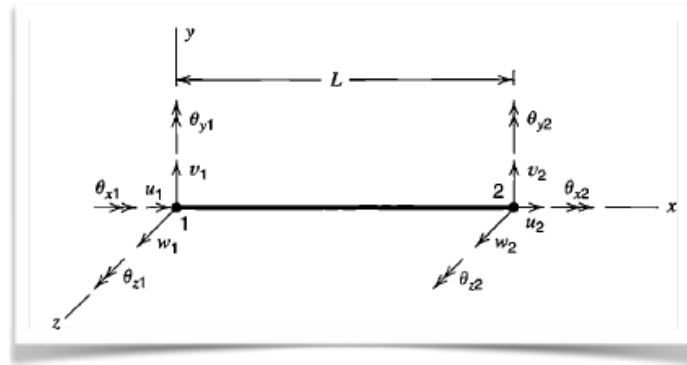


Figure 3.1.2.1. 3D beam element in rectangular coordinate system [Cook et. al, 2001]

To solve a 3D beam Timoshenko theory is applied, which allows for shear deformation, making the model more accurate. 3D beam can be used for modelling almost all structures using simple mesh. The cross-sectional area can have many shapes, but it has to be constant along its length. It requires 4 coordinate systems, global, local, coordinates for internal forces and cross-sectional coordinates. Element is represented as a line, therefore user must assure the cross-section is oriented in the right direction as indicated on Figure 3.1.2.2.

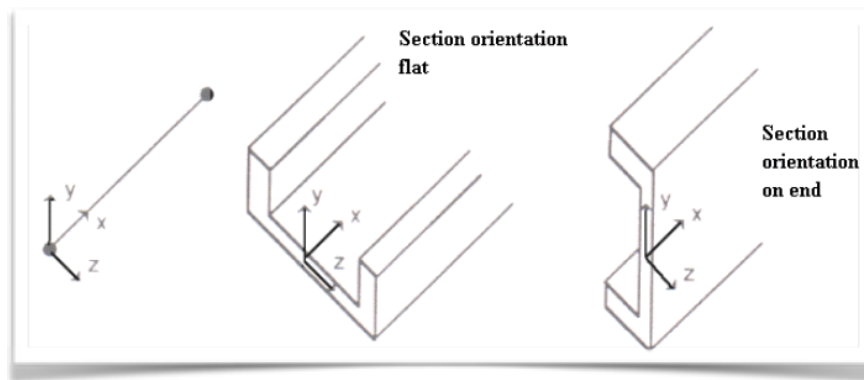


Figure 3.1.2.2. Example of different orientation of beam's cross-section [www.fem-infos.com, 2008]

The loads that beam sustains are:

- Dead load, the self-weight of beam itself
- point loads and moments, on any point of the element
- linearly varying line load
- thermal load, linearly varying, parallel or perpendicular to the beams length
- motion of rigid or elastic support

The results of an analysis are:

- displacements and rotations in global coordinate system

- six internal forces at local coordinate system (bending moments, tensional moments, shear forces, normal force)

An example of model, which uses 3D beam elements can be seen on Figure 3.1.2.3.

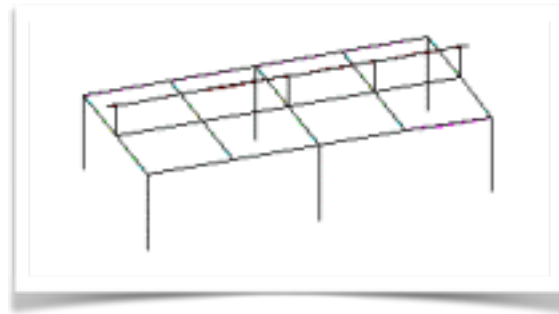


Figure 3.1.2.3. Simple structure modelled using 3D beam elements [www.fem-infos.com,2008]

2D beam

Is similar to 3D beam, it has two nodes, straight axis and is lying in XY plane. Number of degrees of freedom is 3 per node. An example may be seen on Figure 3.1.2.4.

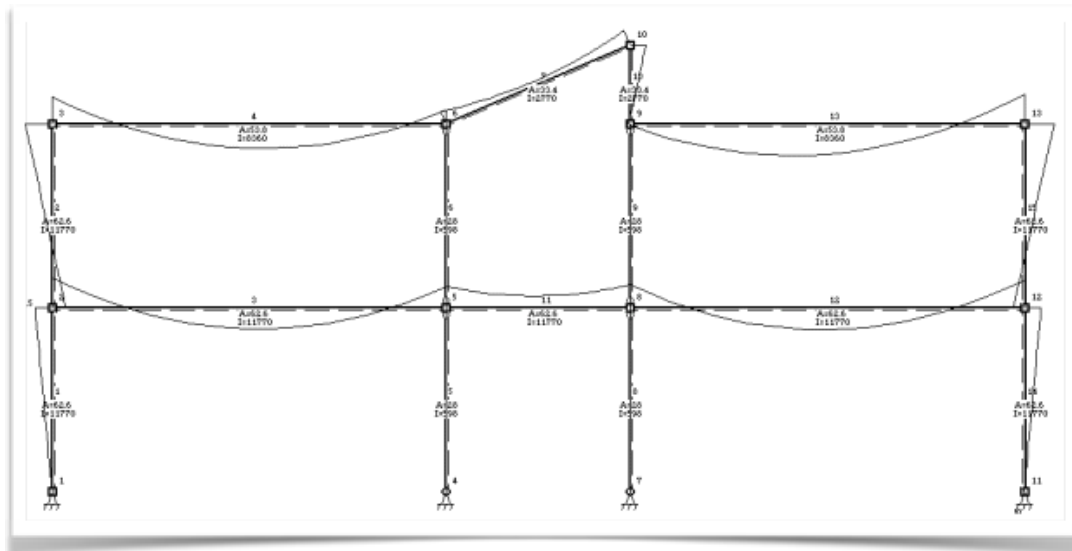


Figure 3.1.2.4. Model done with use of 2D beam elements [www.fem-infos.com,2008]

3D shell

An isoparametric, thick shell element, which can have six or eight nodes as seen on Figure 3.1.2.5. Each node has six degrees of freedom, three displacements and three rotations.

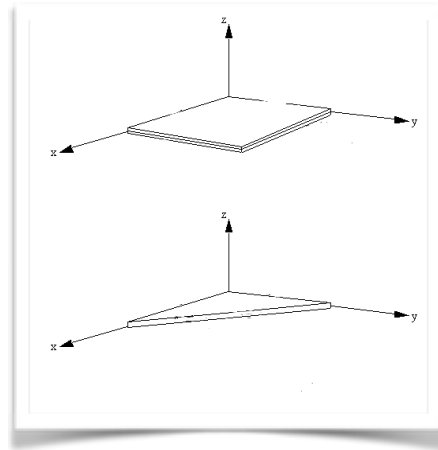


Figure 3.1.2.5. Example of shell elements with eight and six nodes [www.fem-infos.com,2008]

The 3D shell elements can be used for modelling spatial structures, which has one dimension much smaller than other two. They can sustain forces and moments parallel in their plane as well as perpendicular to their plane. Using the thick shell theory, also known as Kirchhoff hypothesis, which is an extension of the Euler-Bernoulli beam theory to thin plates, allows to calculate the cross-sectional shear forces more accurate.

Loads that the shell accounts for:

- self-weight (dead load)
- point forces and moments
- line load
- surface load (pressure)
- thermal load
- motion of supports

Results of the analysis:

- rotations and displacements
- moments, shear forces, normal forces

Example of modelling with 3D shell elements can be seen in Figure 3.1.2.6.

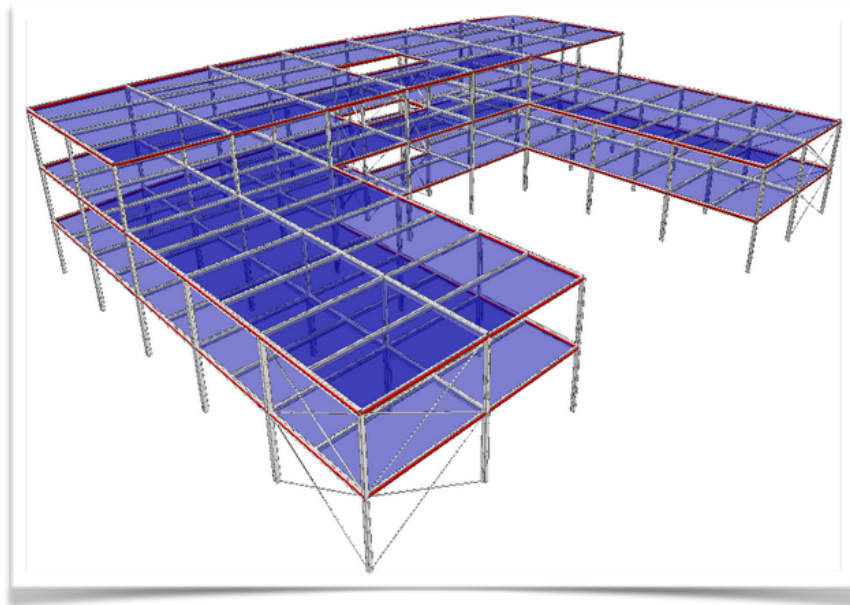


Figure 3.1.2.6. Finite element model using 3D shell elements as slabs [www.masterseries.com, 2014]

2D plate

Plate element is very similar to 3D shell element, but it is able to calculate only bending effects. This element is lying only in the XY plane, loads are perpendicular to its plane. It has 3 degrees of freedom per node, one displacement in Z direction and two rotations around X and Y axis as seen on Figure 3.1.2.7.

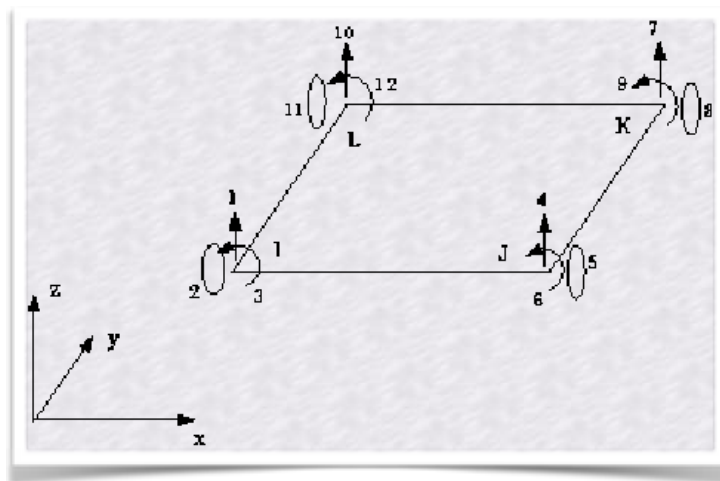


Figure 3.1.2.7. Internal forces on 2D plate element [http://www.isr.umd.edu/, 1996]

Plate is an orthotropic element and can have 3 or 4 nodes and example of modelling with 2D plate, using 3 node elements can be seen on Figure 3.1.2.8. The mesh gets finer in places where is the stress expected to be higher to achieve better results.

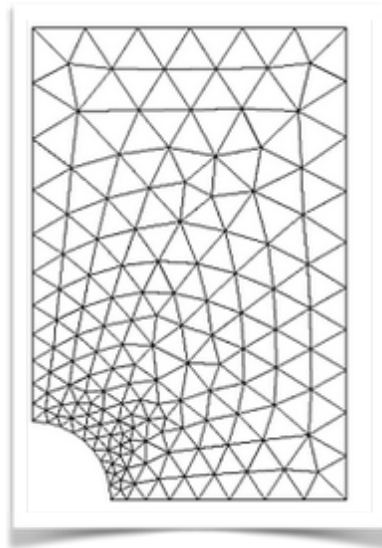


Figure 3.1.2.8. Example of modelling with 2D plate elements [www.cosmol.com, 2013]

2D wall

Wall element has similar properties to 3D shell element, but in this case it is able to calculate only membrane effects. The element is planar, lying in XY plane, loads are also acting in its plane and the material is considered orthotropic. There are only two d.o.f. and these are displacements along the X and Y axis.

3D solid

A 3D solid elements can be used to model almost all volumetric structures. To model a structure would require a great number of these elements, large meshes as can be seen on Figure 3.1.2.9. This demands for increase in computational power and time. In places where the structure is under great stresses the mesh density must be larger to obtain accurate results. Knowing this, other elements like beam or shell should be used when possible.

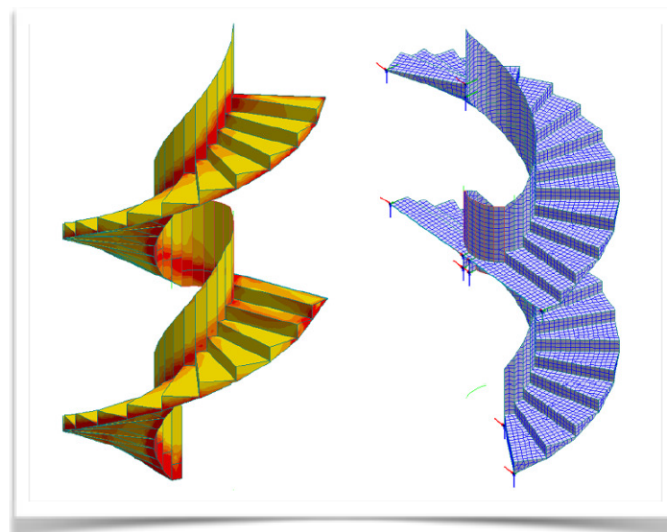


Figure 3.1.2.9. Finite element mesh with the use of 3D solid elements applied on staircase [Kristensen, 2014]

Solid element can have various shapes, for example tetra element has 4 or 10 nodes depending on how accurate the results should be as seen on Figure. 3.1.2.10.

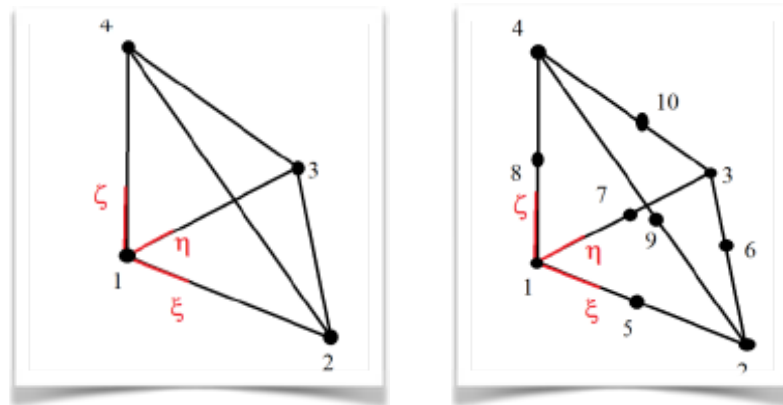


Figure 3.1.2.10. 4 and 10 node tetra element [strusoft.com,2010]

Next is wedge element, which can have 6/18 nodes or brick element having 8/27 nodes similarly as adding nodes to tetra element. Wedge and brick element they can be seen on Figure 3.1.2.11. At each node a solid element has three degrees of freedom and these are the three displacements. For solid elements with 4, 6 or 8 nodes the displacement interpolation is done with the use of linear function, but with elements that have 10/18/27 nodes the second-order polynomial function is used.

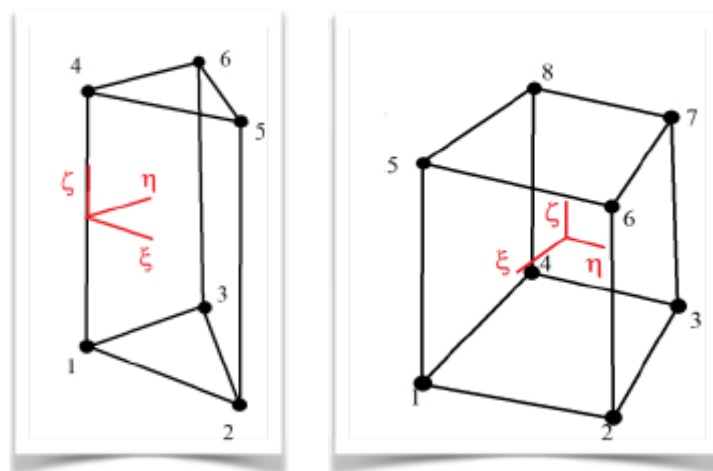


Figure 3.1.2.11. Drawing of wedge and brick elements [strusoft.com,2010]

This problem requires two Cartesian coordinate systems one global and one for element orthotropy, if orthotropic material used. Stress strain relationship is defined by Hook's law.

Loads:

- dead load
- point forces
- line loads

- surface load (pressure)
- thermal load
- shrinkage load
- support motion

Results:

- displacements
- stresses

Point support

Point support is a point element that has 6 degrees of freedom, generally it can be said that it has 3 stiffnesses against rotation and 3 against movement. To have different support types the program modifies the values of these stiffnesses. The outcome are the reaction forces and moments.

Line support

Line support is like elastic support, with 3 nodes as seen on Figure 3.1.2.12. Each node has 6 degrees of freedom, 3 displacements and 3 rotations. The system adjusts the stiffnesses for these d.o.f according to the type of support. Results of this element are reaction forces and moments along its length.

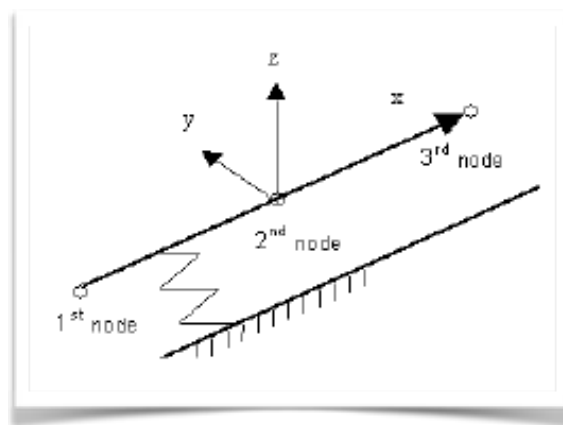


Figure 3.1.2.12. Line support with 3 nodes in local coordinate system [strusoft.com,2010]

3.1.3. Finite Element Mesh

To generate finite element analysis of a structure the model, an example can be seen on Figure 3.1.3.1., has to be discretized into mesh of finite elements. Good mesh size for a structural plate element is between 0,5 meter up to 1,2 meters depending on the type of calculation that will be executed, smaller mesh is usually not necessary. It would only cause increase in calculation time. For structural elements as walls mesh between 0,2 to 0,5 meters recommended the program will try to get these values as well depending on the settings. Both plates and walls are divided into finite shell elements. For structural line elements like beams and columns it is recommended to use 4-6 finite elements for concrete structures. If there is truss structure, like roof, this can be increased up to 20.

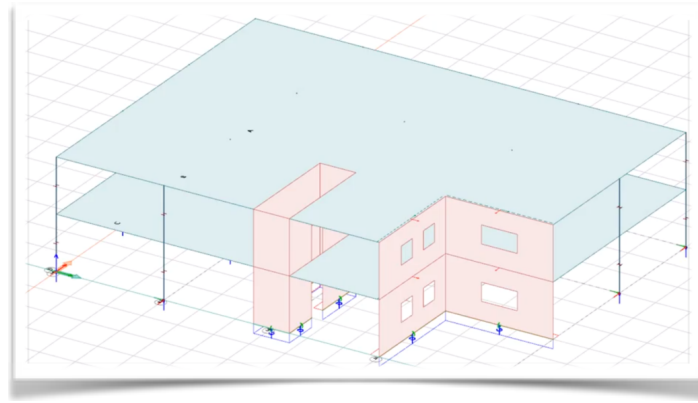


Figure 3.1.3.1. Example of a structural model in FEM-design

Generating mesh

In FEM-design the tool, for creating mesh of a selected objects, breaks down the structure into sub-regions and generates multi-phased mesh.

Phases for mesh generation:

- Defining vertices of elements

Vertices or nodes of the elements are placed apart from each other at a distance of the size of an average element following lines which are parallel with the longest side of the sub-region. These parallel lines are as well distant by the average size of an element. An example of a sub-region and created mesh can be seen on Figure 3.1.3.2.

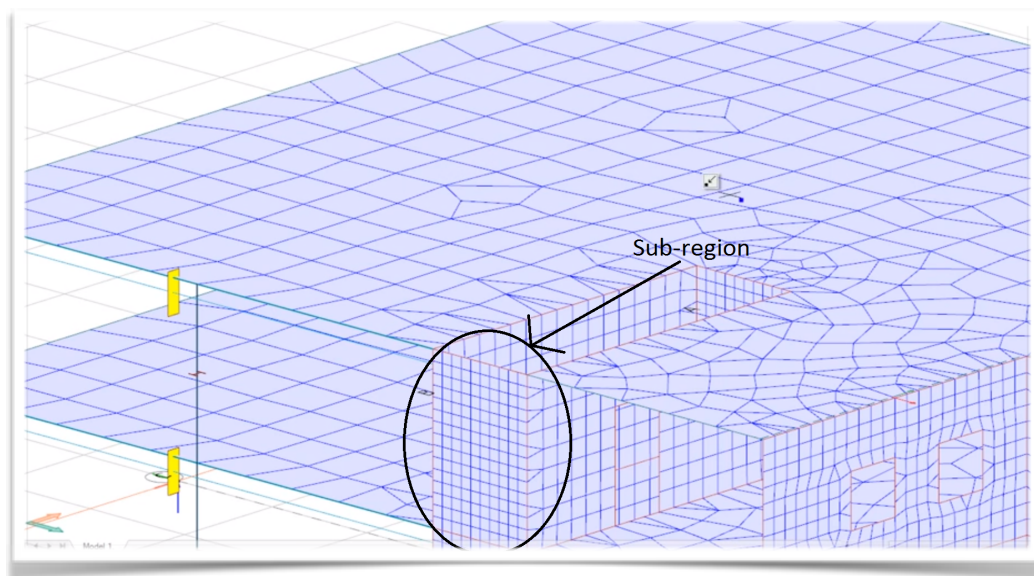


Figure 3.1.3.2. Example of a sub-region in FEM-design, where the mesh size changes

- Creating triangle mesh using vertices

Triangular mesh is created using Delaunay triangulation technique with use of Voronoi domains. The

principle is shown on Figure 3.1.3.3.

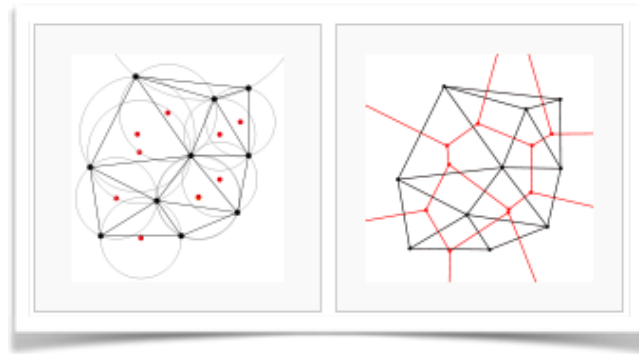


Figure 3.1.3.3. Delaunay triangulation, with circumferences and their centres and Voronoi diagram connecting the centres [wikipedia, 2013]

- Converting mesh of triangular elements into mix of quadrate-triangle mesh

This function converts the triangle mesh into globally optimal shapes of mixed quadrate-triangle mesh.

- Optimising the coordinates of the nodes (Smoothing the mesh)

Smoothing the mesh is done with use of a procedure, which selects places for nodes of triangles in a way that the area of triangles is balanced. Iteration technique is used to achieve this.

- Setting middle points to sides of elements

At the sides of each element a mid-point node is created.

Mesh generating tool in FEM-design possesses few automatic features. If the model requires it the software will make mesh in some areas thicker, where higher stresses are expected. Generated mesh can be seen on Figure 3.1.3.4.

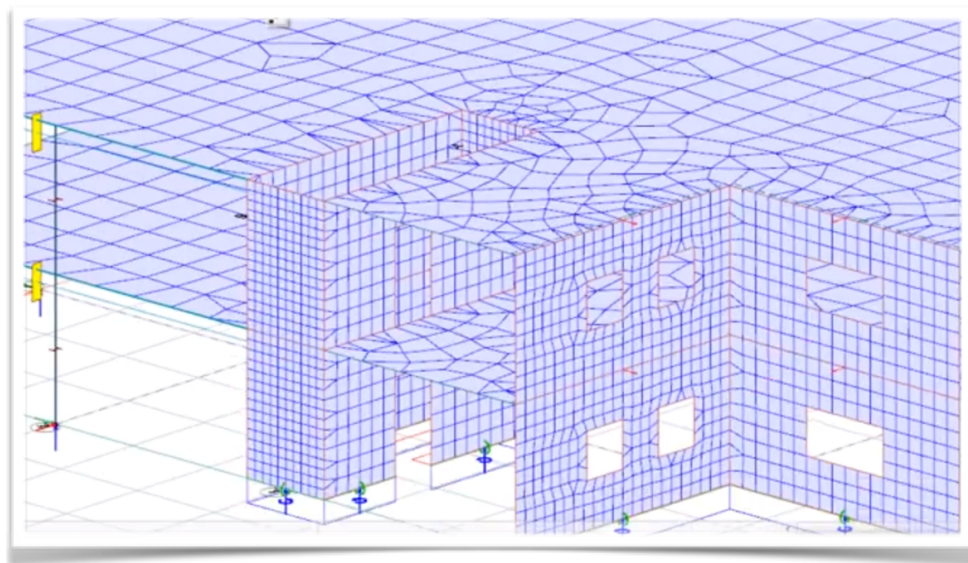


Figure 3.1.3.4. Example of change in mesh density

When the program finds such place number of iterations will be greater than 1. If the required density

could not be achieved, warning will appear. This means there is an anomaly in the model. Another automatic help is done when the pre-set elements size is too large, in this case the program will redo the meshing and reduce element size. When is the program generating the mesh for certain sub-region, where the mesh is for example finer, it takes into account the borders of this region with a different sub-region if these have nonidentical mesh, the program regenerates the mesh in the corresponding border as can be seen on Figure 3.1.3.5. This also increases the number of sub-regions. If the recalculation of mesh is done too many time there could be problem with the model, if the mesh is too dense it always indicates critical parts of the structure's model.

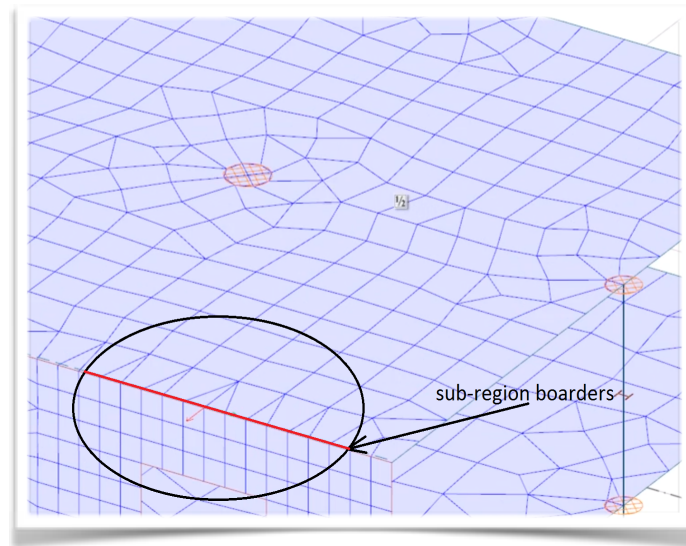


Figure 3.1.3.5. Bordering sub-regions

Refining mesh

Refine tool is used to create denser mesh. The user can choose to set certain critical places where the stress is expected to be higher to have denser mesh. This can be in point or line supports and loads, borders of regions with different material, element end points, element intersection and connections. As can be seen on Figure 3.1.3.4. the mesh was set to be denser at column end points.

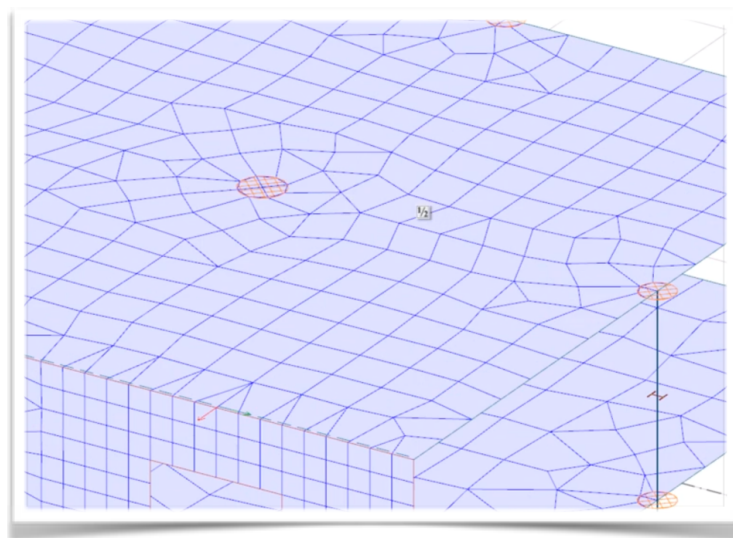


Figure 3.1.3.4. Example of change in mesh density, to achieve more exact solution

3.1.4. Load Group Combinations

After having the structure modelled, divided into finite elements loads can be applied to complete the analysis. To do this user should create load groups. These should be defined by different duration classes like permanent, seismic, temporary or accidental. Every load group contains one or more load cases as dead load/ self weigh of the structure, live load, wind load, etc. Different partial safety factors can be assigned to the load cases depending on limit state that is being created. The software will check every possible combination and find the least favourable one. Permanent loads will be presented at all load combinations. The limit states involve Ultimate Limit State (ULS) and Serviceability Limit State (SLS).

3.2. Design

FEM-design gives the possibility to calculate suitable reinforcing of concrete elements, design of steel and timber structures. The design can be made automatically by the software or made manually by the engineer.

4. Testing

Now that a peek inside the theory behind FEM-design was presented, testing of the software will follow. This chapter is going to use a sample building model, which will be created in FEM-design, divided into finite elements and after applying loads the analysis will be carried out. Results will be displayed. After having the model set and ready the connection and transport of the structural model from FEM-design into Revit is going to take place in the next chapter.

4.1. Creating Model

First step is to create a model of the sample structure. The model can be seen on Figure 4.1.1. Each floor is identical, therefore the easiest way is to model the first story and copy the rest.

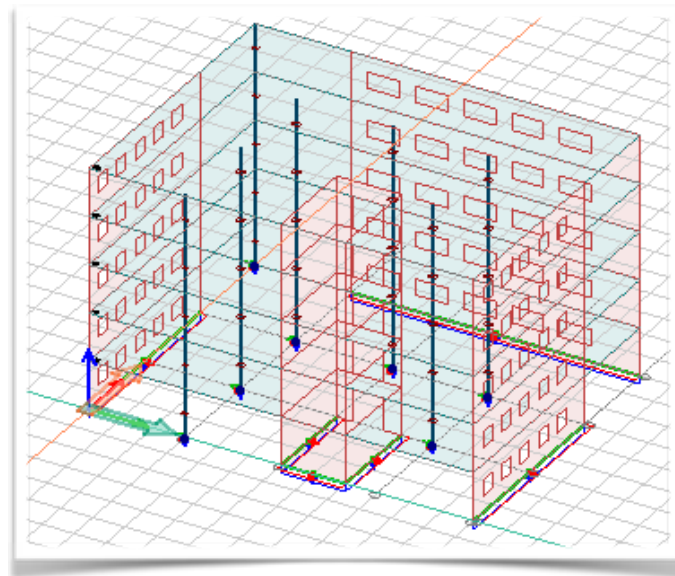


Figure 4.1.1. Sample building

When filtering the model different structural components, stories, analytical elements (point or line supports), components with certain material type or cross section can be viewed separately. Selecting the columns only will leave a clear view as seen on Figure 4.1.2.

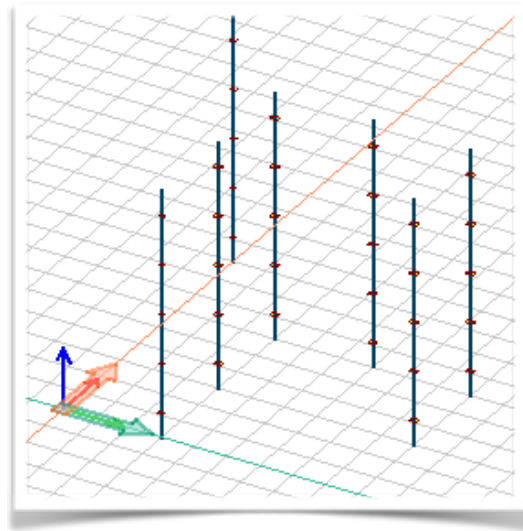


Figure 4.1.2. Filtering columns

By pointing at each column material type and cross section appears. To create a column a structure tab is opened and from bar like elements column element is chosen, the height is set to 3 meters, placement is chosen *Above* as seen on Figure 4.1.3.

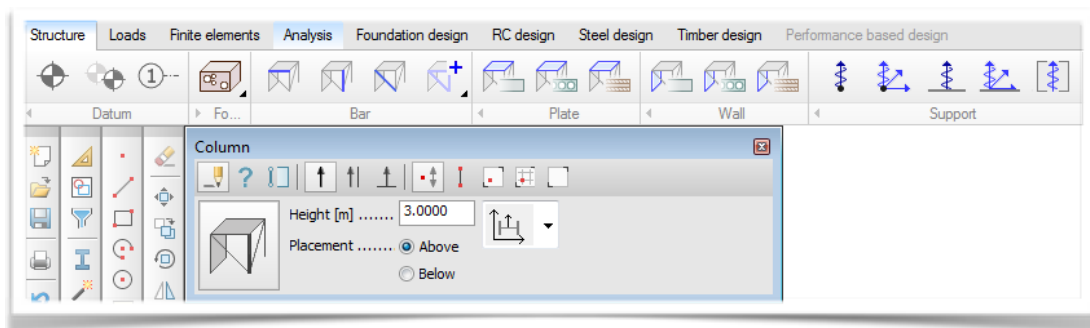


Figure 4.1.3. Creating column element

The model consists of two steel S 235 columns of cross-section HE-A 300 and five concrete C30/37 columns with rectangular 300x450 cross-section. Clicking on *Default Settings* the cross-section and material is chosen as seen on Figure 4.1.4.

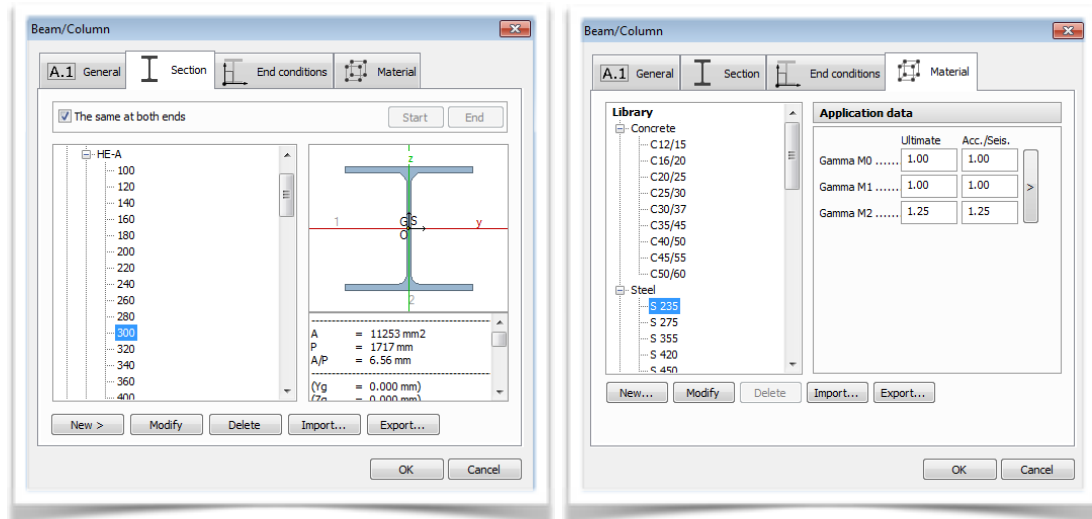


Figure 4.1.4. Setting material and cross-section of steel column

Next the walls and plates are created by using the plane wall and plane plate tool, the material is set to C25/30 and thickness to 0.2 meter. The floor can be divided by axis, which help to select a favourable point of view, in this case to create a window opening as can be seen on Figure 4.1.5.

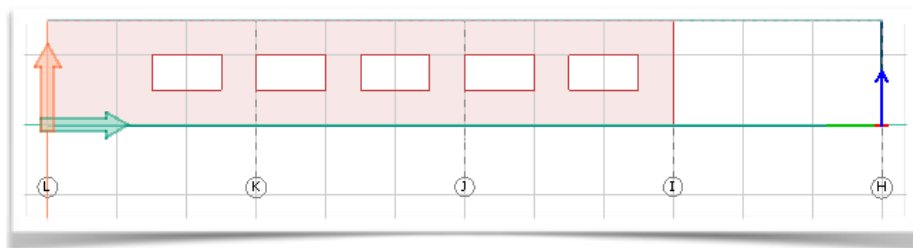


Figure 4.1.5. Cross-sectional view

The Story tool can be also used for creating new stories, changing height of the story, for copying story or deleting it. Therefore now that the first floor is done the rest can be copied as can be seen on Figure 4.1.6. The supports are created as rigid point supports for columns and rigid line supports for walls.

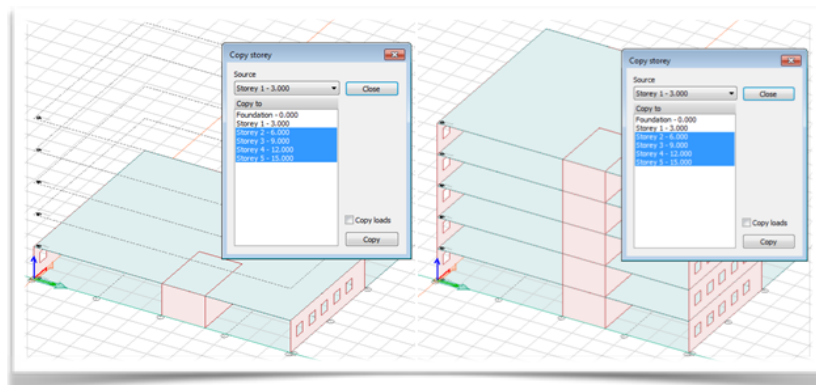


Figure 4.1.6. Copying stories

4.2 Applying Loads

When having the model created and supports set, the next step is adding loads to the structural system. Loads are in FEM-design defined as load cases, each load case has one or more load objects. Examples of different load cases can be seen on Figure 4.2.1. Type of load case set to ordinary means that no additional effect is assigned to the load, having shrinkage as type of load will add additional shrinkage strain behaviour as an effect in concrete design.

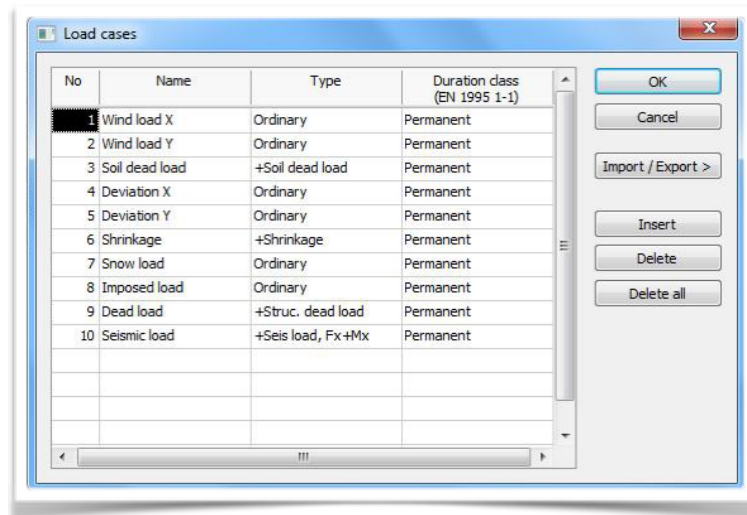


Figure 4.2.1. Example of different load cases

The dead load or self weight of the structure is calculated automatically by the program based on geometry and material properties of the structural elements. Load combinations are created by combining load cases and multiplying them with load factors. The types of load combinations that can be defined are ultimate (U), accidental (U_a), seismic (U_s) or serviceability limit state. This project will use two ULS and SLS. ULS is defined to calculate strength and stability and SLS calculates displacements. The applied loads were, the dead load of the structure, live load, snow load and wind load in X and Y direction as can be seen on Figure 4.2.2.

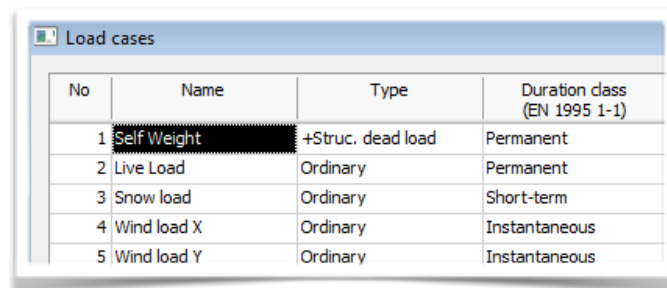


Figure 4.2.2. Applied load cases

Live load is applied on every floor area with the magnitude of 10kN/m^2 , snow load is applied only on the roof with the magnitude of $1,6\text{kN/m}^2$. Wind load is calculated according to Eurocode by selecting the wind speed, building height and type of terrain as can be seen on Figure 4.2.3. The dead load is calculated automatically by the program.

Wind speed [m/s] 26.0

Building height [m] 15.0

Terrain type:

- ☐ 0. Sea or coastal area exposed to the open sea
- ☐ I. Lakes or flat and horizontal area with negligible vegetation and without obstacles
- ☒ II. Area with low vegetation such as grass and isolated obstacles (trees, buildings) with separations of at least 20 obstacle heights
- ☐ III. Area with regular cover of vegetation or buildings or with isolated obstacles with separations of maximum 20 obstacle heights (such as villages, suburban terrain, permanent forest)
- ☐ IV. Area in which at least 15 % of the surface is covered with buildings and their average height exceeds 15 m

Storey	Level [m]	X+ [kN/m]	Y+ [kN/m]	X- [kN/m]	Y- [kN/m]
Storey 1	3.00	4.09	3.81	4.09	3.81
Storey 2	6.00	4.09	3.81	4.09	3.81
Storey 3	9.00	4.09	3.81	4.09	3.81
Storey 4	12.0	4.09	3.81	4.09	3.81
Storey 5	15.0	2.04	1.91	2.04	1.91

OK Cancel

Figure 4.2.3. Applying wind load in accordance to Eurocode

Having set the load types and their magnitudes the load combinations are created. As can be seen on Figure 4.2.4, five load combinations are set with different load factors.

No	Name	Type	Factor	Included load cases
1	LC1	S	1.00	Self Weight
			1.00	Snow load
			1.00	Live Load
			1.00	Wind load X
2	LC2	U	1.00	Self Weight
			1.00	Snow load
			1.00	Live Load
			1.50	Wind load X
3	LC3	U	1.00	Self Weight
			1.00	Snow load
			1.00	Live Load
			1.50	Wind load Y
4	LC4	U	1.00	Self Weight
			1.00	Snow load
			1.50	Live Load
			0.75	Wind load X
5	LC5	U	1.00	Self Weight
			1.00	Snow load
			1.50	Live Load
			0.75	Wind load Y

OK Cancel

Import / Export >

Load combination

Generate

Insert

Copy

Delete

Delete all

Load case

Insert

New

Remove

Figure 4.2.4. Load combinations

4.3 Design and Analysis

Going into analysis window the results of the analysis can be displayed. Now that all the calculations ran the steel design and concrete design are checked if the load bearing capacity of the elements is sufficient. Having only two steel columns fast calculation is ran and the results are shown, both columns are satisfying the design as can be seen on Figure 4.3.1.

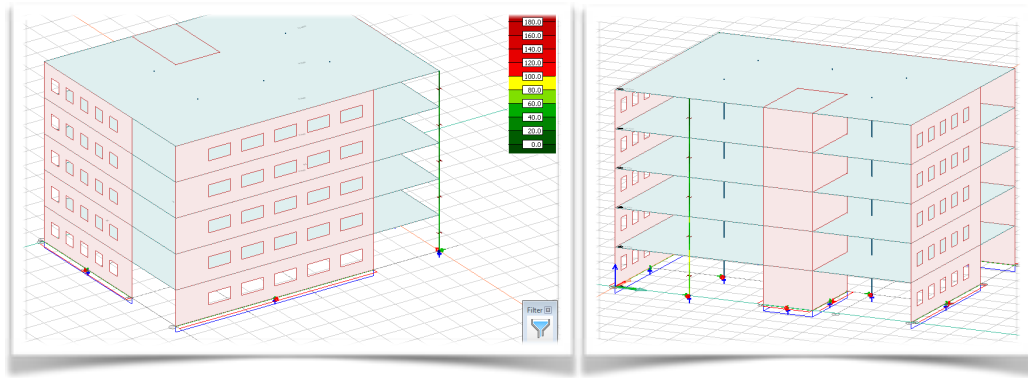


Figure 4.3.1 Steel columns utilisation

Going into detailed results and clicking on an element the analysis calculations is revised. Shear and torsional resistance, flexural buckling, torsional-flexural buckling, lateral-flexural buckling and interaction between normal forces and bending. At the end summary is viewed. Summary results of HE-A 300 steel beam column can be seen on Figure 4.3.2.

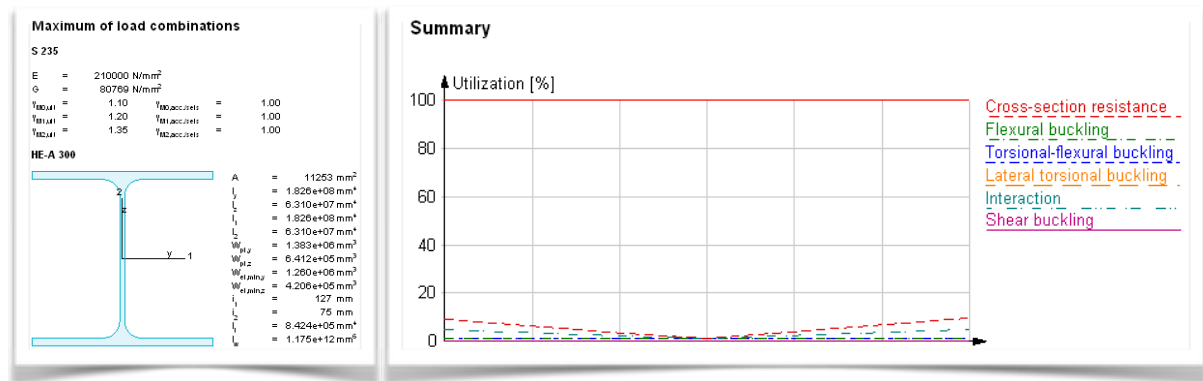


Figure 4.3.2. The cross-section of steel column and calculation summary

Tab with RC design is opened and a auto design of all structural elements is performed. Focusing first on the concrete columns, the cross-section of columns are designed to 500x400 mm and material type to C30/37, now a suitable reinforcement is found as can be seen of Figure 4.3.3.

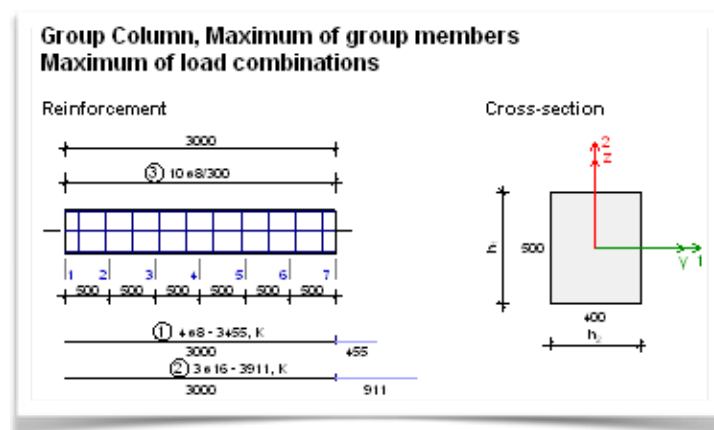


Figure 4.3.3. Column reinforcement design

Detailed results show the applied reinforcement, material properties, data about the cross-section, calculation formulas, table of results, stress strain graphs, utilisation graphs. Utilisation of reinforced columns is visualised by showing results of RC columns as seen on Figure 4.3.4.

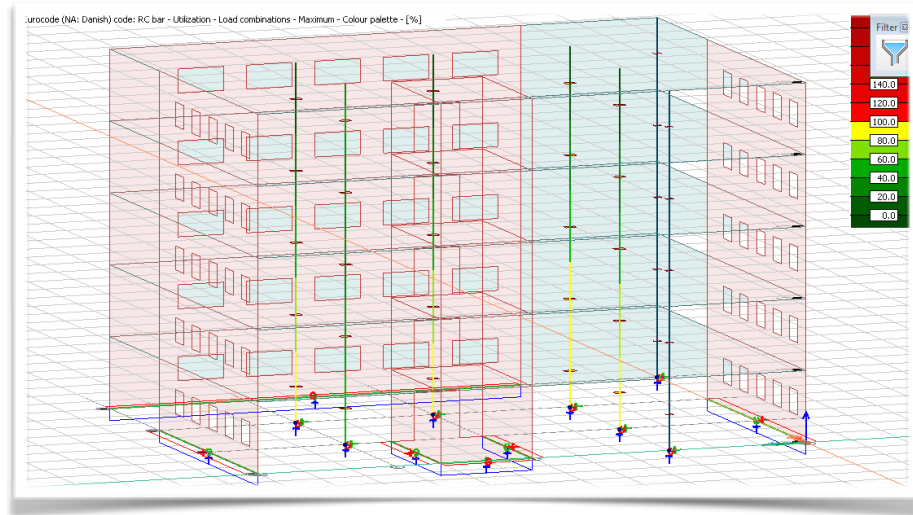


Figure 4.3.4. RC column utilisation, maximum load combination

Second the surface reinforcement is designed involving the plates and walls. As in column elements come of the properties of elements had to be changed to arrive at the suitable reinforcement. On Figure 4.3.5. the top reinforcement layer in x-direction for shell elements is shown.

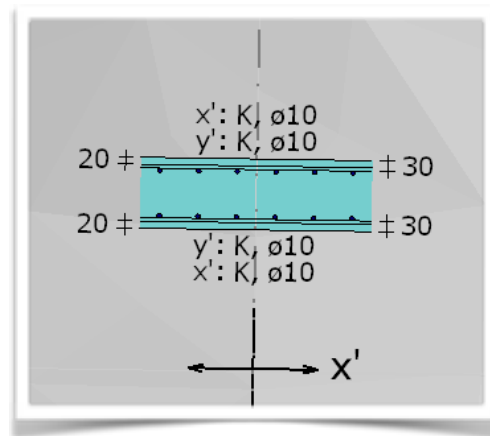


Figure 4.3.5. Top reinforcement layer x-direction

4.4 Finite Element Mesh

Few settings are changed before the finite element mesh is generated, firstly auto peak smoothing around the end of columns is selected, element size is set to be calculated region by region and default minimum element number for bar element should be set to 5. Resulting finite element mesh is shown on Figure 4.4.1.

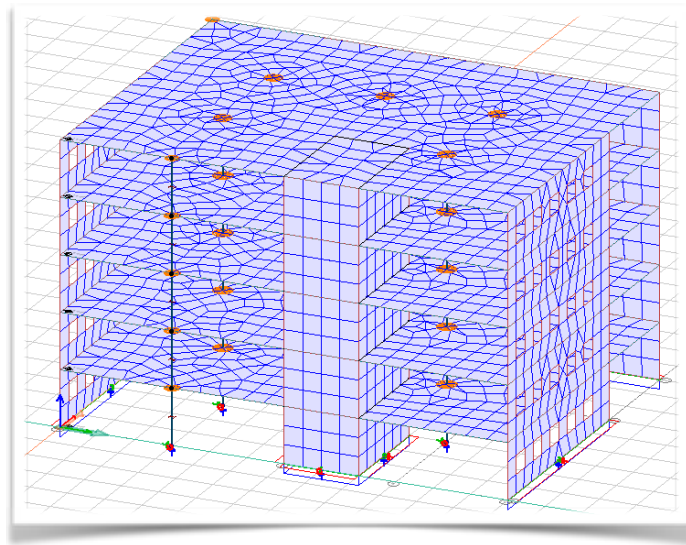


Figure 4.4.1. Finite element mesh of sample building

4.5 Results

After running the analysis and arriving to satisfactory design the results can be displayed. For example the displacements are visualised on Figure 4.5.1.

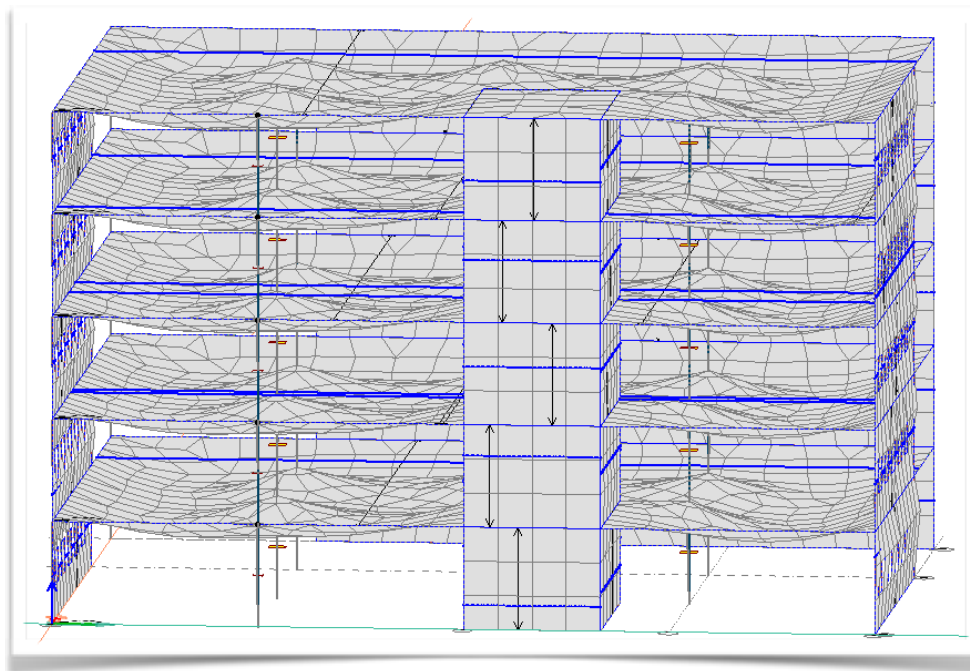


Figure 4.5.1. Results of analysis-displacement

In the next chapter the transfer procedure between Revit and FEM-design is going to be reviewed and tested.

5. Revit to FEM-design Connection

When having a model in Revit ready the transfer can be done using StruSoft add-in. First some points on how this transfer works will be made and later the actual process will be described. The information exchange is based on transfer of the analytical model by saving the file in xml format, called struxml. The add-in makes possible to transfer the model's geometry and some of its parameters, which are common in both softwares. This transfer works both ways, any model created or updated in Revit can be exported as struxml file and opened in FEM-design and vice versa. However the transfer is based on analytical model data exchange, therefore if a structural element in Revit does not have enabled analytical model it cannot be transferred. This needs to be checked before the data transfer is launched and will be illustrated later. Non-structural elements can not be assigned as analytical. All structural elements as columns, floors, walls, framing elements etc. can be marked in Revit as analytical and therefore transferred back and forth as can be seen on Figure 5.1.

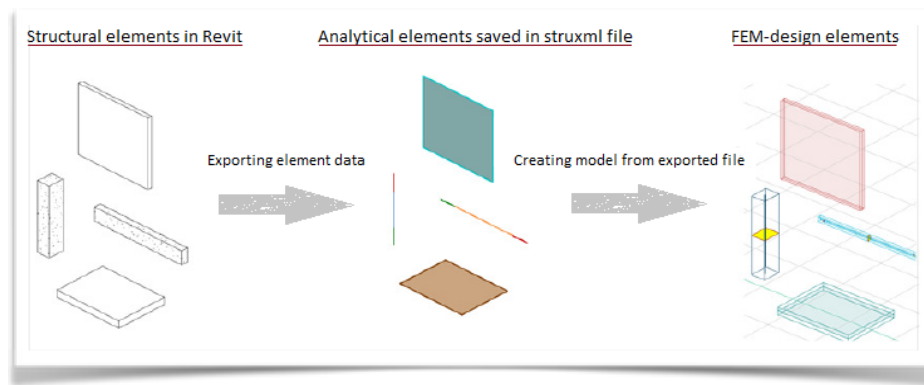


Figure 5.1. Export of elements through analytical model

The information transfer between Revit and FEM-design is illustrated on Figure 5.2. It is possible to export the entire model from Revit to FEM-design or make only changes in a model that already exists as indicated by the two yellow boxes on the figure. For example, an architect is designing a building using Autodesk Revit, creating a complex model with elements that have load-bearing purposes as well as with non-structural elements and properties such as covering materials, colours, furniture, etc. Now the load-bearing system needs to be designed by a structural engineer, in order to withstand the loads it might experience along its lifespan. Therefore, the architect or the structural engineer in charge of the structural model exports the analytical model from Revit, using the StruSoft add-in export function, as the smallest arrow marked with II. shows on Figure 5.2. This saves the model in a special type of file, struxml. This file possesses information about the geometry of the elements, their materials, support axes and levels set on the model. The file can be sent and then opened in FEM-design by a structural engineer. Here the system is designed and checked as a load-bearing system. Designing reinforcement for concrete elements and steel elements might change geometry and some properties of the entities. In this case, the changes need to go back to the original design in Revit, which means the import tool will be used, shown on Figure 5.2, by a blue box marked with 5. This cycle continues until the optimal design is reached.

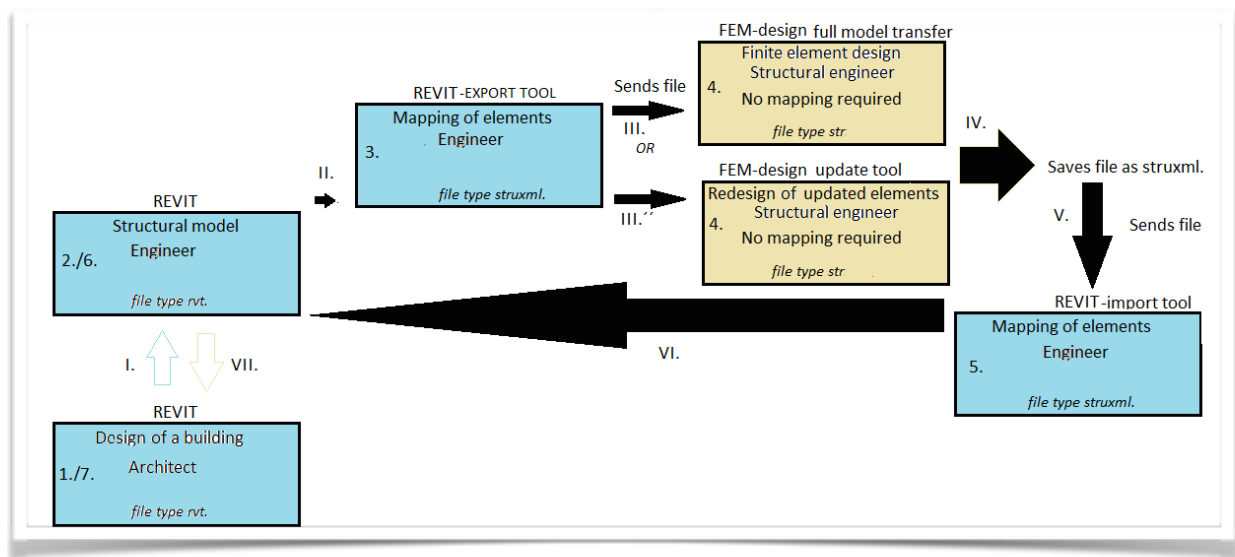


Figure 5.2. Information flow between Autodesk REVIT and StruSoft FEM-Design

5.1 Basic Principles of Import and Export

Some of the basic principals of the transfer are mentioned in this section and later the transfer itself will be tested. The model can be transferred back and forth with add-in by StruSoft. The information exchange is based on transfer of the analytical model by saving the file in xml format, called struxml. This transfer works both ways, any model created or updated in Revit can be exported as struxml file and opened in FEM-design and vice versa. This process requires mapping of the elements wish to be transferred. Before mapping a list of elements that are going to be transferred needs to be made, because when mapping some element properties from current model will have to be matched manually. This can be e.g. cross-sections of bar elements, thickness of shell elements or their materials. If needed this properties have to find its pair in the collaborating software.

The following elements and properties can be transferred using StruSoft add-in.

Elements:

- Datum elements (axes, storeys)
- Bar elements (beams, columns, trusses)
- Shell elements (plates, walls)
- Profiled plate panels (imported as plates)
- Foundation elements
- Supports
- Loads and load cases

Properties:

- Material of an element
- Section /Profile /Thickness of an element
- Releases of linear elements
- Boundary conditions
- Eccentricity in case of floor slabs, walls and beams

Datum elements

Datum elements are storeys and axes. One of the requirements for importing model to Revit is having at least one story set as Revit needs to know at what elevation to create a model. Stories from FEM-design are transported as levels, without keeping the original name given in FEM-design, just as level 1, level 2, etc. Axes are transported as grids, the exact length and geometry is kept, but names are not copied. They will be counted and named in order of creation in FEM-design.

Bar elements

Every cross-section and material of a bar element wish to be transferred has to be mapped separately after loading a model. When importing to Revit corresponding types need to be found in libraries loaded into the project or created and then while mapping selected to match its element model.

Shell elements

Each of shell element has to be mapped separately, finding the corresponding type for each floor or wall. In this case the material and geometry of the element is selected at the same time while mapping.

Supports

The exact supports and support groups can be transferred.

Loads and load cases

Load types as point load, line load and surface load are available for export/import.

5.2 Workflow

In this section the work flow from Figure 5.2. is investigated and the model transfer is tested. The transfer will start from exporting a created model from Revit into FEM-design.

Exporting Revit model to FEM-design 3.

If it is time to do the structural analysis of the building, the architect or structural engineer, handling the CAD model made in Revit, can launch the transported. The data transfer is done via StruSoft export tool as Figure 5.2. indicates by the smallest arrow marked with number II. The structural model created in Revit can be seen on Figure 5.2.1.

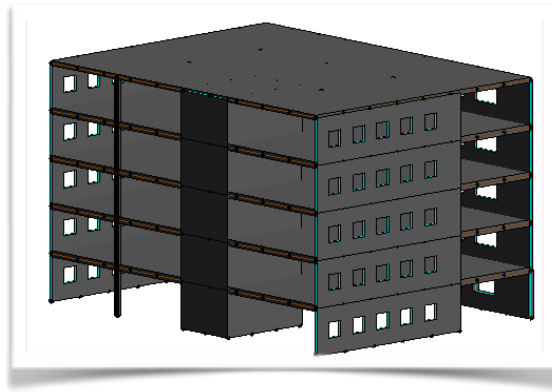


Figure 5.2.1. Physical model of sample structure in Revit

First all the elements wish to be transferred need to have enabled analytical model as mentioned before. By clicking on an element appears description of the entity along with its properties, scrolling down the structural properties are checked if the analytical model is enabled. Figure 5.2.2. shows a structural wall with enabled analytical model.

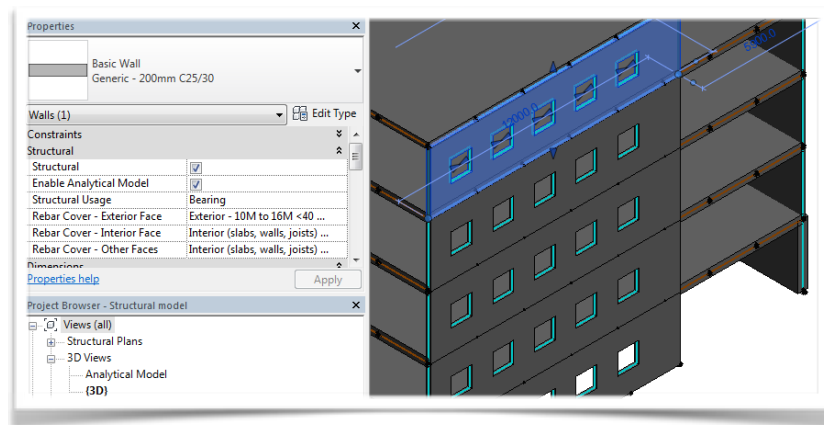


Figure 5.2.2. Structural wall with enabled analytical model in Revit

The whole analytical model can be seen on Figure 5.2.3.

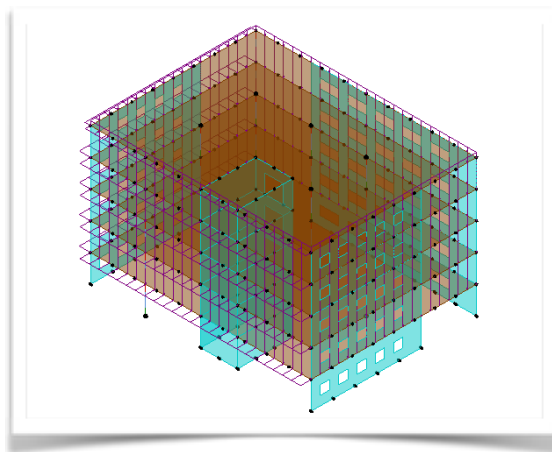


Figure 5.2.3. Analytical model displayed in Revit

Now through StruSoft add-in the engineer can launch the transfer of the model through the export tool as shown on the Figure 5.2. by blue box with number 3. A mapping window is opened as can be seen on Figure 5.2.4. below.

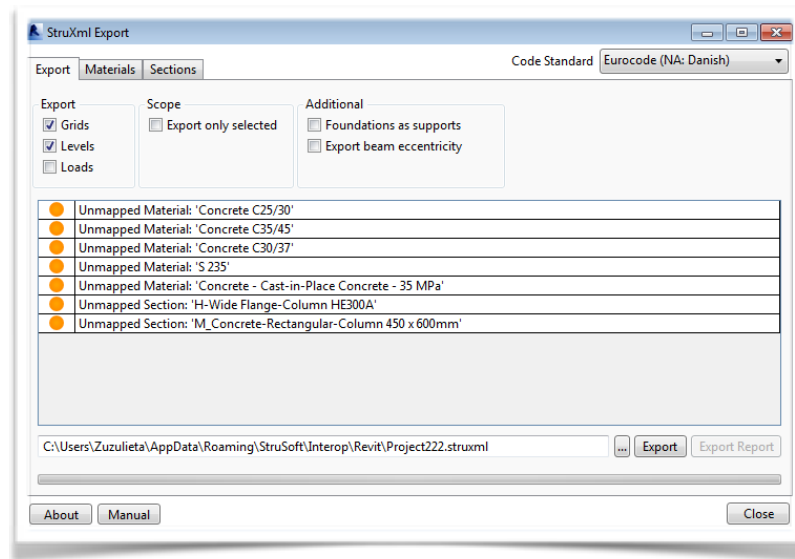


Figure 5.2.4. Mapping of element properties for export from Revit

The window indicates all the element properties that need to be mapped. In the additional windows as materials and sections the necessary properties need to find their match in the collaborating software. The materials window is opened and can be seen on Figure 5.2.5. On the left side there are material properties of the elements from Revit that are wish to be transferred. On the right side there is the library from FEM-design where the corresponding type is chosen from.

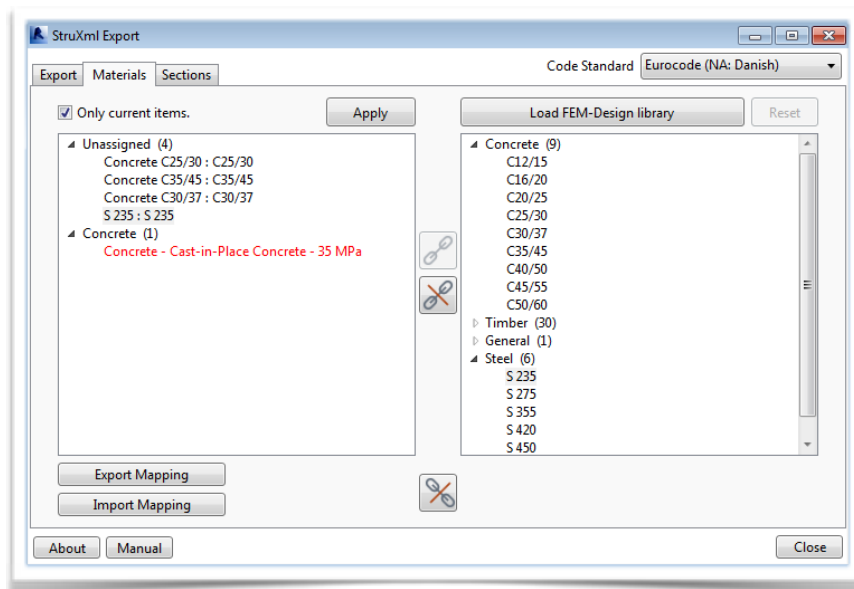


Figure 5.2.5. Material mapping before exporting the analytical model

This is the manual work that the export tool requires. The mapping is easy as all the section types and materials are already listed in the FEM-design library. However one of the materials from Revit is missed placed. *Cast-in-place concrete* as seen on the Figure 5.2.5. was not intended to be used, this is a problem facing this mapping case. Closing the mapping window and trying to find an element with this material property led nowhere. Chosen material to match was C35/45 as this was the one with closest physical parameters. All the sections were found with no problems. Now that all the element properties have their match the model is exported into struxml file, saved and can be e.g. send by email to the structural engineer for analysis in FEM-design this is indicated on Figure 5.2. by arrow marked with III.

Managing a model in FEM-design 4.

After structural engineer receives a struxml file, it can be simply opened in the software FEM-design this is shown on Figure 5.2. by a top yellow box marked with 4. Running analysis and designing steel and concrete elements may lead to changes in geometry or material properties of the model. In this case the analysis led to a error of a wall in the first story as seen of Figure 5.2.6. This means that the optimal reinforcement could not be found as the maximum utilisation was exceeded.



	Shell	Max. [%]
⊖	W.7.1	106

Figure 5.2.6. Warning while designing reinforcement, maximum utilisation was exceeded

In such case a geometrical or material property of the wall need to be changed. W.7.1. has thickness of 400mm and materials property set to C35/45. The thickness was changed to 500 millimetres and the calculation was ran again. Now the suitable reinforcing was designed with no problems. An additional change was done on the model, walls were added as can be seen on Figure 5.2.7.

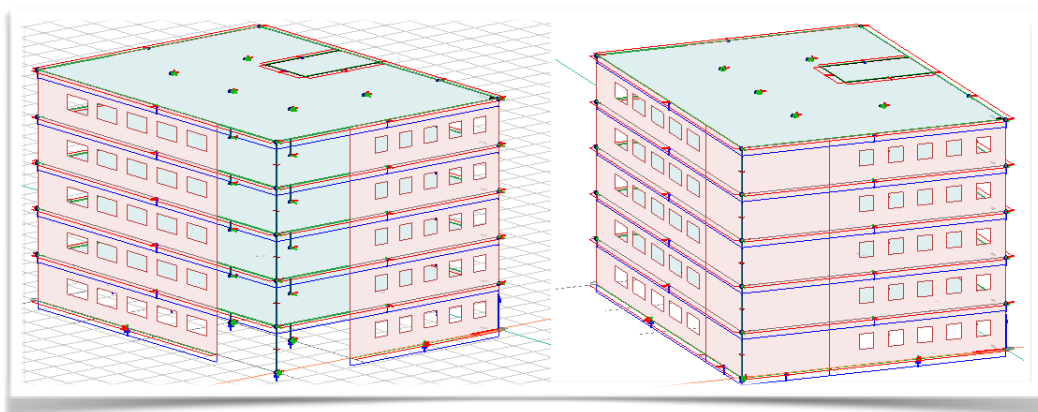


Figure 5.2.7. Design changes made in FEM-design before the transport

Now the changed structural model has to go back to the engineer and architect that handle the CAD design in Revit, where it will be updated. In the introduction on Figure 1.2. is shown that the update tool should work both ways, however currently this tool is not active, therefore the import tool has to be used as shown on Figure 5.2. by blue box marked with 5. Structural engineer needs to save the entire model

into xml file and send it back to Revit as indicated on Figure 5.2. by arrow IV. and V. Here a new model will be received. This is not the best solution as the architectural model is complex model composed also of non-structural elements and designs as can be seen of Figure 5.2.8.

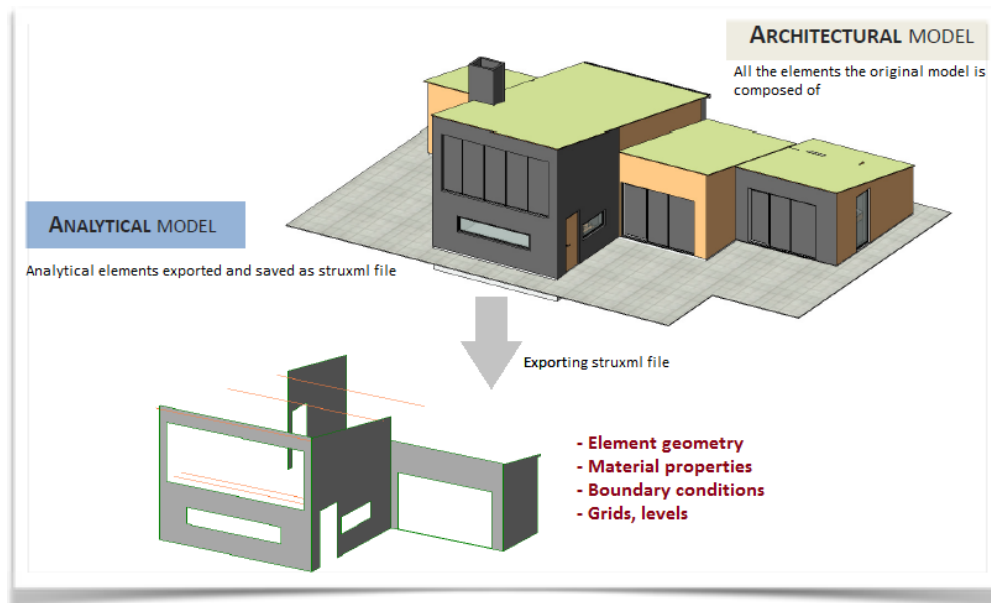


Figure 5.2.8. Analytical model vs architectural model

Therefore during the time that the structural engineer is working on analysis and design of the load-bearing system, the architect could have done some other work on the CAD model in Revit. By importing the entire model back to Revit a new project is created and any changes that were made before would be lost or they would have to be saved in another project. If the update tool would be active and used, both disciplines could be working at the same time, and only the load bearing structural system would be updated in the architect's design. The update tool works in the opposite direction though, from Revit to FEM-design. If the architect changes properties of the structural system, only the updates can be made in a FEM-design model. This option will be investigated at last as the workflow would go on. Since the Revit update does not work the import tool is used and discussed below. The model currently in FEM-design is saved in struxml file and send back as indicated in Figure 5.2. by the arrows marked with IV. and V.

Importing model into Revit 5.

The data exchange from FEM-design to Revit requires that all the elements, that wish to be transferred exist in Revit project previous to the transfer. Therefore if the model is originally made in Revit some of the elements are already inside. Otherwise before the transfer begins all the element types that are required to be mapped have to be listed and loaded from Revit libraries into the project. If the library does not have the specific types, the elements have to be created inside the opened Revit project. After loading several libraries, still many element types were not found, therefore they had to be created manually. Having to create all the needed sections, materials etc. inside Revit project, before mapping the actual element type is bit inconvenient and time consuming. Creating them could also cause errors. For example while the creation process the engineer gives the element a name and the physical

properties are assigned inside edit tool in the element's description as seen on Figure 5.2.9. This means that if a property given to the element is misplaced the element still has the right name. Later finding a n element with a erroneous property is tedious work.

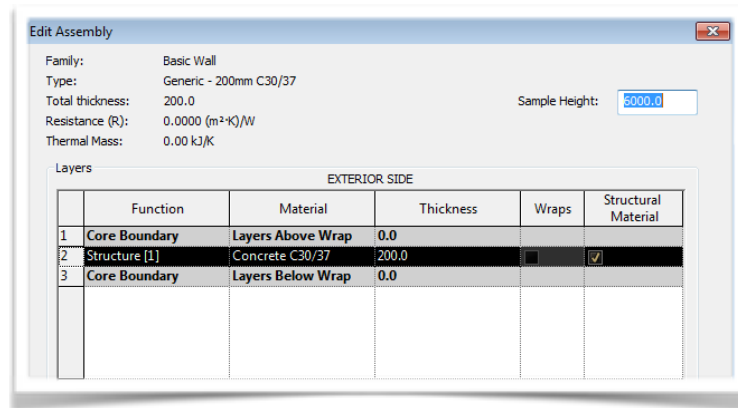


Figure 5.2.9. Creating wall element for import of a model

In the case of shell elements, the corresponding wall or slab is assigned with specific structural material as this is required while the mapping of shell elements, the creation is seen on Figure 5.2.9. For bar elements the cross-section and material type is matched separately. When all the elements that need to be mapped are loaded inside the Revit project the mapping can start, the import is shown on Figure 5.2. by blue box marked by number 5. A window opens, where all the element types in the model are listed as seen of Figure 5.2.10.

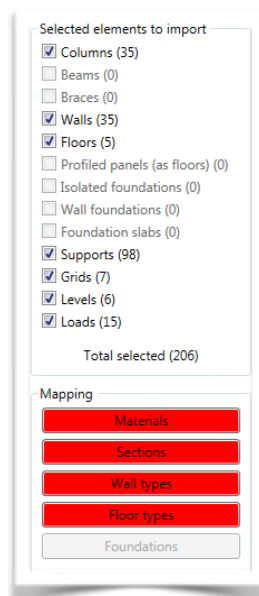


Figure 5.2.10. Mapping summary of elements being imported

As Figure 5.2.10. indicates that he supports, grids, levels and loads are transferred automatically by the software. As seen on the figure it is possible to deselect element type that is not desired to be transferred.

The mapping of column section is presented on Figure 5.2.11., on the left side there are the specific sections of columns needed to be matched from FEM-design and on the right side, there are the corresponding ones found in Revit.

Column mapping	
FD Sections	Revit Sections
Concrete sections: Rectangle; 400x500	Structural Columns: Precast-Rectangular Column; 400x500mm
Steel sections: HE-A; 300	Structural Columns: H-Wide Flange-Column; HE300A

Figure 5.2.11. Mapping the cross-sections of columns being imported

Finished mapping of wall type elements can be seen on Figure 5.2.12. As mentioned before, geometry of shell elements are mapped together with their materials also visible on the figure below.

Wall type mapping	
FD Types	Revit Types
concrete: C25/30 - 200 mm	Walls: Generic - 200mm C25/30
concrete: C30/37 - 200 mm	Walls: Generic - 200mm C30/37
concrete: C30/37 - 300 mm	Walls: Generic - 300mm C30/37
concrete: C35/45 - 400 mm	Walls: Generic - 400 mm C35/45
concrete: C35/45 - 500 mm	Walls: Generic - 500 mm C35/45 2

Figure 5.2.12. Mapped wall elements

After the mapping is done import of the elements is completed by report sheet, all elements were imported without error. The changed model in Revit is displayed on Figure 5.2.13., where also can be seen the added walls.

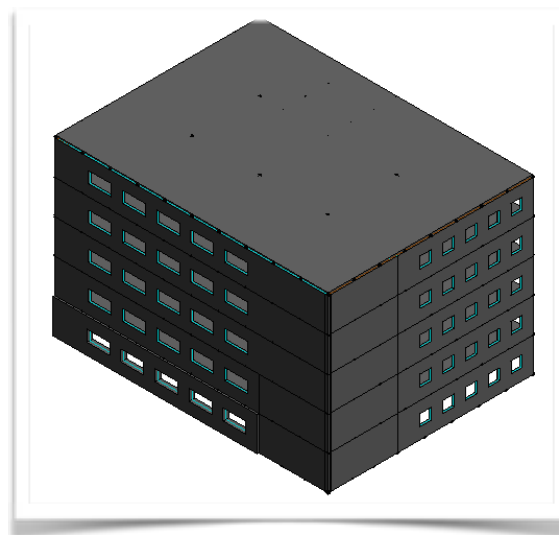


Figure 5.2.13. Imported model after changes, Revit

There is a possibility to save mapping, which is convenient. This means that creating and mapping of most of the elements is done only once. As this is a model building given by StruSoft it was expected that

the import would go smoothly. However in some cases there can be issues with having more complicated models and errors while transport. Now that the model is back in Revit, this time when updates are made on the design the update tool can be used, which is shown on Figure 5.2. by the arrow III." and bottom yellow box. This tool will transfer only the changes made on the model, which will save time as the whole mapping process was done already and saved.

Updating FEM-design model 4.

Update tool enables the engineer to make changes in his project without having to create new model, shown on Figure 5.2. by bottom yellow box marked with 4. For example a beam dimension is changed by the architect in the CAD model, the reinforcing therefore needs to be recalculated as well. By using the update tool, the structural engineer is able to update his model without creating an entire new case and the analysis could be ran again immediately. This is convenient, because even though the loads are transported along the way when using the export tool the load combinations are lost. Therefore if the engineer wants to run a new analysis using export tool the load combinations would have to be recreated. It is convenient to use the update tool, which keeps all the settings made in FEM-design previously and changes only the updates on the structural model. The update should change only the new added, changed or deleted components. It is essential to always save Revit model after the first model is exported. This gives the model a identification number that will help to keep track of the updated files. While updating the coordinates have to stay the same, if not the process will not work. Updatable entities window will open where all the elements that where changed appear. The sign plus indicated added elements, minus deleted entities and arrow updated ones as can be seen on Figure 5.2.14.

Entity ID	FD-entity's date	↔	Imported entity's date
Bar: "C.2.1"	21. 3. 2015 16:24:34	-	-
Bar: "C.10.1"	21. 3. 2015 16:25:34	-	-
Bar: "C.4.1"	21. 3. 2015 16:25:34	-	-
Bar: "C.6.1"	21. 3. 2015 16:25:34	-	-
Bar: "C.8.1"	21. 3. 2015 16:25:34	-	-
Shell: "W.4.1"	19. 3. 2015 21:32:45	»	7. 6. 2015 20:11:02
Shell: "W.6.1"	19. 3. 2015 21:35:30	»	7. 6. 2015 20:11:02
Shell: "W.3.1"	21. 3. 2015 15:50:05	»	7. 6. 2015 20:11:02
Point support group: "S.11"	15. 5. 2015 17:34:52	»	7. 6. 2015 20:10:34
Point support group: "S.12"	15. 5. 2015 17:34:54	»	7. 6. 2015 20:10:34
Point support group: "S.13"	15. 5. 2015 17:38:04	»	7. 6. 2015 20:10:34
Point support group: "S.15"	15. 5. 2015 17:38:48	»	7. 6. 2015 20:10:34

Figure 5.2.14. Update summary

On the left there is displayed the type of element ready to be updated, next there is the date the entity was last modified in FEM-design and on right the date that the element was updated in the importing file. This way the engineer can choose which updates to make. Updates in Revit model were made and these are, the material of a first floor slab was changed from C30/37 to C35/45, extending of walls and deleting a front column. After the update is completed the changes can be seen on Figure 5.2.15. the walls were added, the column was deleted along with its supports and also the material was changed.

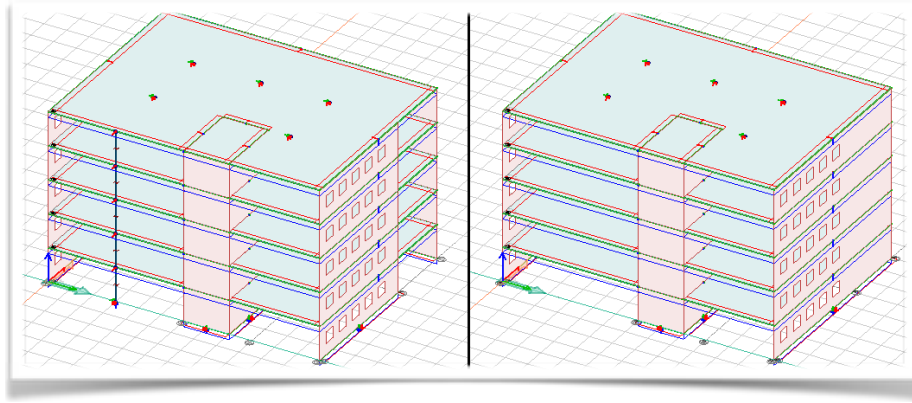


Figure 5.2.15. Original FEM-design model and model after updates, deleted column and added walls

Having construction project usually includes many changes and updates along the process, therefore the collaboration between the two most important practices needs to be handled with care. Even small changes should be updated as soon as possible. Also if having a big model it would be unnecessary and time consuming to export or import and map the entire model every time. In this case the update tool is helpful. When going back to Revit with any changes made in FEM-design, the update tool is not available. Therefore the process of import will have to be repeated. However it is possible to save mapping therefore the long initial process of loading and creating elements in Revit does not have to be repeated and only the new or changed elements are mapped.

6. Conclusion

This report investigated the possible automatization of information transfer between architectural model and finite element analysis. While creating a structure, disciplines handling designs separately in different softwares need to collaborate in the most effective way. Classical way should be replaced by modern data transfer using information technologies. These allow to abandon the manual work, which is time consuming and could cause errors and make the transport of models automatic. As the process is still quite new on the market, several challenges can be faced. This project analysed the information transfer between Autodesk Revit handling the design of a structure and FEM-design software handling the finite element analysis of the load-bearing system. This data transport is possible through StruSoft Add-in that enables the transport of analytical model of a structural model through xml file. Processes of the transport were investigated and analysed throughout the report.

The classical way of collaboration between CAD model and finite element analysis is by 2D drawings of a structure. This makes the process time consuming and open to errors as mostly manual work is required. StruSoft add-in makes this process by a part automatic. The process of exporting a model from Revit to FEM-design by StruSoft add-in already handles the geometry of the structure. The information from the struxml file carry the coordinates of each element, its material type, and dimensional properties, etc. some of this can be seen on Figure 6.1.

```

</bar>
<bar type="column" guid="b1ae12bb-4069-4f35-acff-e211f3cd4fcb" last_change="2015-03-21T17:25:34.000" action="added">
  <bar_part guid="9ca08d9f-558e-4021-9b2b-5fc1484e6c80" last_change="2015-03-21T17:25:34.000" action="added">
    name="C. 9.1" complex_material="8219874b-1ee4-4ca2-a5d8-d2b57d6827b1" made="rolled" ecc_cal="true"
    complex_section="a0eb6ca1-2d59-44cc-b580-31a4fb6fd524">
      <curve type="line">
        <point x="0" y="18" z="12"></point>
        <point x="0" y="18" z="15"></point>
      </curve>
      <local-y x="0" y="1" z="0"></local-y>
      <connectivity m_x="true" m_y="true" m_z="true" r_x="true" r_y="true" r_z="true"></connectivity>
      <connectivity m_x="true" m_y="true" m_z="true" r_x="true" r_y="true" r_z="true"></connectivity>
    </bar_part>
  </end>
</end>

```

Figure 6.1 Code of analytical model data of a bar element saved in struxml file, opened in notepad

As can be seen on the figure above the code carries information about the time that the last changes were made on the bar element, the type of the bar element, the name, its material, type of cross-section, geometry, coordinates and end connections. The only thing that needs to be done manually is to find matching properties in FEM-design for some of the element types, being transferred from the Revit model. This means if the transport includes 10 concrete columns having the cross-section 300x300 the corresponding cross-section needs to be found in FEM-design library and mapped together, the same accounts for material type. Element types as supports, loads, grids and stories are transported automatically. Going the reversed way, when importing a model into Revit from FEM-design the model also holds information about the reinforcement of concrete elements. This time if transporting an element type that is not inside the opened Revit project means that it has to be loaded or created inside the Revit model before the transfer is launched. For example if the analytical model being imported involves a concrete C 35/45 wall with thickness of 600mm and this exact type of wall is not present in the Revit model it has to be loaded or created. Loading libraries did not help to find most of the elements, therefore they had to be created. This can be a little inconvenient when having too many various elements. Creating them is the manual work required for the transfer to work.

The process of changing a model until arriving at a suitable design is repeated many times. Therefore the transport of the entire model is unnecessary. Even though the mapping of the elements can be saved and only new entities should be matched and created when using import/export tool, other problems are faced. When running finite element analysis the load groups and combinations are created and settings regarding finite element mesh are made, however when importing new model, even though the loads could be carried along the analytical model, these settings would be lost. When working in FEM-design the update tool can be used, which keeps the setting made in the model and only the changes of newly created or updated elements are made in the project. One more obstacle can be faced, if an element is deleted in FEM-design, but not in Revit and the update tool is used the element may reappear in the FEM-design model. This is possible to avoid by deselecting this element from the list of entities being updated. Also the update tool is not active when one wants to update a model in Revit by importing changed model from FEM-design. Therefore no changes in the Revit model can be done while the finite element analysis is ran and calculations are made. These would be lost after the import.

This process could be helpful for collaboration and avoiding errors in the design, but is not without disadvantages. Including it into an ongoing workflow, could be a challenge as this add-in works only for the connection between FEM-design and Revit or Tekla software. Although FEM-design has quite

familiar and intuitive interface, which would save time if learning more complicated and completely new tools in a different finite element software. Many tutorials are available with support from the developing team and can be used while learning. Future implementation of automatisation of the data transport into workflow is necessary and should be thought about. This process will be improved in time and much beneficial for the working process.

7. Bibliography

- kebmin.dk, 2014.** Jasper Caruso, *Danish government to strengthen construction sector*, URL: <http://www.kebmin.dk/en/news/danish-government-to-strengthen-construction-sector> , 7-11-2014, Seen: 15-2-2015.
- nationalbimstandard.org, 2015.** National institute of Building Science, *About the National BIM standard*, URL: <http://www.nationalbimstandard.org/about.php>, 2015, Seen: 15-2-2015.
- Ling, Xue-yuan, Xi-la, 2011.** Qin Ling, Deng Xue-yuan and Liu Xi-la, *Industry foundation classes based integration of architectural design and structural analysis*, Shanghai Jiaotong University and Springer-Verlag Berlin Heidelberg, 2011.
- Cook et al., 2001.** Robert D. Cook, David S Malkus, Michael E. Plesha and Robert J. Witt, *Concepts and applications of finite element analysis*. ISBN-13: 978-0-471-35605-9, Hand book. Wiley, 2001.
- colorado.edu, 2006.** Dr. H. Qi, *Finite Element Analysis*, Colorado University, URL: http://www.colorado.edu/MCEN/MCEN4173/chap_01.pdf, 2006, Seen: 28-2-2015.
- strusoft.com, 2010.** StruSoft, *FEM-design Applied Theory and Design*, 2010, Downloaded: 8-3-2015.
- www.fem-infos.com, 2008.** FEM infos, *Element types of Finite Element system*, URL: http://www.fem-infos.com/Element_types/Element_types_of_FEM_system_MEANS_V9.html, 2008, Seen: 28-2-2015.
- www.masterseries.com, 2014.** www.masterseries.com, *Finite Element Stiff Deck*, URL: <http://www.masterseries.com/blog/tag/finite-element/>, 12-5-2014, Seen: 28-3-2015.
- www.isr.umd.edu, 1996.** Mark Austin, *Library of Finite Elements*, URL: <https://www.isr.umd.edu/~austin/aladdin.d/fe-library.html>, 1996, Seen: 28-3-2015.
- www.cosmol.com, 2013.** Walter Frei, *Meshing Considerations for Linear Static Problems*, URL: <http://www.cosmol.com/blogs/meshing-considerations-linear-static-problems/>, 22-10-2013, Seen: 28-3-2015.
- Kristensen, 2014.** Denis Kristensen, *StruSoft Program Universe*, StruSoft FEM-design presentation, 2014.
- wikipedia, 2013.** Wikipedia, *Delaunay triangulation*, URL: http://en.wikipedia.org/wiki/Delaunay_triangulation, 28-10-2013, Seen 30-3-2015.