LAB University of Applied Sciences
Lappeenranta
DDCIV20
Double Degree in Civil and Construction Engineering

Artem Shatalin

Reinforcement design using SOFiSTiK in Revit

LAB University of Applied Sciences

Thesis 2020

Abstract

Artem Shatalin
Reinforcement design using SOFiSTiK in Revit, 60 pages, 4 appendices
LAB University of Applied Sciences
Lappeenranta
DDCIV20
Double Degree in Civil and Construction Engineering
Thesis 2020
Instructors: Lecturer Timo Lehtoviita, LAB University of Applied Sciences;
Department Manager Olli M. Salo, AFRY; Structural Designer Sander Vaher, AFRY
Consultants: Ehsan Ghazanfari, AFRY; Dmitry Sokorev, AFRY; Dmitry Yaroshutin, SPBSUACE; Andrei Yashanov, apex projet bureau; Olga Pertseva, PSS GRAITEC.

The main scope of the research is to explore usage of Design + Analysis, Reinforcement Generation, and Reinforcement detailing plug-ins by SOFiSTiK on Revit for underground structures. Discovering software limitations and ways how these could be handled. Work contains also elements of BIM implementation and automatization methods in the commissioning company.

The study was commissioned by AFRY, Department of Rock Engineering and Underground Spaces. Research was tested on real-case example of maintenance tunnel of 130m in length, with curved geometry and consisted of 10 construction sections.

Finding the most efficient way of reinforcement design was done using following tools: Revit used for modelling as a pre-processor, SOFiSTiK for FEA, and design of reinforcement. According to calculation results reinforcement is being modelled in Revit. The outcome is reinforcement detailing drawings for concrete structures, calculation report, BIM model, and rebar schedules.

Recommended modelling methods are provided by SOFiSTiK webinars and guidelines, basic FEA logic, user reports from conferences, and personal experience of author.

Based on the findings, it can be concluded that Revit + SOFiSTiK workflow can be used for all stages of design project, including all features of BIM approach and FEM analysis. Modelling tools provided by Autodesk and SOFiSTiK in Revit are sufficient to make analytical analytical model for 95% of practical cases. Latter 5% with difficult curved geometries can be handled in native pre-processors SOFiPLUS or TEDDY. Automatization of modelling might be performed using Dynamo visual programming.

Advantage of tested workflow method is simplification of FEA usage for structural engineers, and avoidance of drafting works by possibility of performing the full workflow without Autocad by a single person / engineers team.

Keywords: Revit plug-in, Reinforcement design, Reinforcement detailing, SOFiSTiK, BIM, Structural, FEM, FEA

Table of contents

List of symbols and definitions

BIM – Building information modelling
3D – Three Dimensional (x, y, z)
4D – Four Dimensional (x, y, z, time)
5D – Five Dimensinal (x, y, z, time, cost)
CAD – Competer aidid design
FEM – Finite Elemental Method
FE mesh – Finite Elemental mesh.
FEA – Finite Elemental Analysis
DoF – Degree of Freedom
UI – User Interface
HASE – processor for SOFiSTiK software, allowing to do calculations of geotechnical task with non-linear model of ground
IFC – Open BIM format which was developed by BuildingSmart to unite projects made in different software in one common format
DWG – format for Autodesk AutoCAD. Mostly used for 2D drawings, and more rarely for 3D faces, surfaces, and solid objects.
HVAC – Heating, ventilation and air conditioning.
Physical model – full 3D representation of structural model.
Analytical model – a representation of the structural physical model consisting of analytical elements, geometry, boundary conditions, material properties, and loads. Used for export to generate FE mesh.
Composite model – structural model consisting of physical model and analytical model combined together.
Formwork model – physical model of concrete structure, settled up in right coordinates but not consisting modelled reinforcement.

1 Preface

This Thesis work is written from June to November 2020 for the Department of Rock Engineering and Underground Spaces, AFRY.

The following research is aiming to find ways of replacement of SOFiPLUS and TEDDY pre-processors for SOFiSTiK FEA with addons on Revit and research of its limitations and advantages. According to this method, a new BIM-based workflow for purposes of concrete structures design will be tested for all stages of project from initiation to reinforcement detailing drawings.

The reason of exploring new design workflow is separate data flows in traditional way of design, when there is a need to build separate analytical model for FEA software that cannot be used for further steps after analysis. Combining design process into one line expect to increase quality of design, speed and accuracy.

Thesis includes 7 chapters, with theoretical background in BIM and FEM in chapter two, three and four. In chapter five the workflow shown on real-case examples with testing software capabilities and comparing of results with theoretical data from reference sources. Chapter six and seven consist results and analyze of following research and conclusion how this method can be used in real projects.

In appendixes listed bar bending schedule and drawing for construction block 2 that made only by using Reinforcement Detailing plug-in tools. Also IFC models are available that might be seen by any free IFC viewer such as Trimble Connect or Solibri Anywhere.

I want to thank you my department for support and help during the thesis work – especially my supervisors Olli Salo and Sander Vaher for recommendations in SOFiSTiK FEA features, and Dmitry Sokorev for advising reinforcement design principles and features of cast-in-situ concrete construction in Finland.

2 Introducton

2.1 Background, topic and significance

Design methods are constantly changing during the last century. Firstly, blueprints came to help engineers and architects as the first method of coping drawings. First CAD programs in the middle of the 20th century were not sufficient enough, hadn't such flexibility that was possibly by manual drawing. But it was developing with a time, and now it used to be considered as the most traditional method of producing documentation.

A new step of evolution various design software comes with Building Information Modelling. The significant difference in that method is not only a 3rd dimension for modelling but actually the information, that now our model has. It could be geometry information, cost, weight, manufacturer, stage of construction or any other comments for each element of the structure. BIM in the design stage should make the process of design more efficient, flexible and transparent. To reduce the collisions and mistakes on the construction stage.

BIM approach became more popular and in some cases even necessary type of workflow. But with appearance of more dimensions for the design, we faced new difficulties. Moreover, there is was a lack of time and finances doing the double work without connection between 3D modelling tool, drawings producing, and calculation processes. All those stages were usually performed separately and every time was need to put all initial data from the scratch for each of those tasks. So even if building information model is done, there is still a need to unite this process in a single workflow.

Tekla Structures is a good decision for structural disciplines, allowing flexible modelling of structures in 3D space including reinforcement and especially steel detailing having a rich library of components. But export of Tekla model to FEM software might be problematic, and losses of data might occur because there is no official transfer support and analytical model tracking from the official developer. SOFiSTiK and Robot were used for FEM analysis in the company, but it was needed to use native pre-processors in those programs to model the analytic model. Because this was separate detached workflow and basically it takes the same amount of time to make accurate analytic model, as physical BIM model.

Although, it is possible to merge other disciplines there via DWG or IFC format, there is no way of modifying toposurface, or creating it from other sources. But in fact, in infrastructure design we often need to deal with piping systems and change topography of the area.

Drawings are created as separate DWG file, and there is no way for real-time adjustment from drawing sheets on the model. This way we get it fully detached from model. Traditional BIM workflow shown on Figure 1.

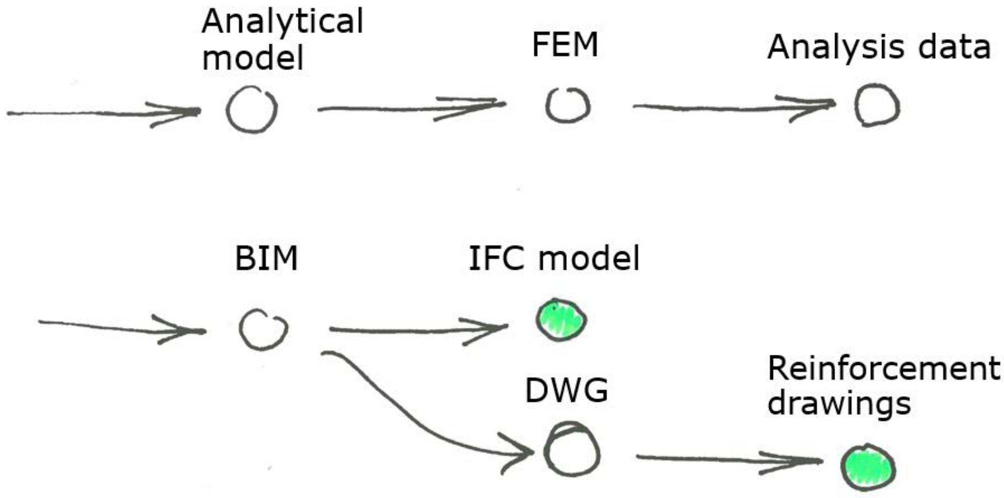


Figure 1. Traditional BIM workflow

According to Bew-Richards BIM Maturity Model, from publication “BIM maturity”, by 4 Jan 2015 and shown on Figure 2 - the level 2 is existing now in a company, where thesis research and implementation is done. Level 2 – is a “file-based collaboration and library management stage”. That means that we have BIM models from different disciplines, we can transfer data between participants, and merge all data in a separate software for BIM collaboration such as Trimble Connect, Solibri, Navisworks, etc. 4D and 5D also might be done on this stage since it’s not the level, it’s an additional dimension of a structure elements.

The next BIM integration level – Level 3. That consist of ideas common data environment, for all project participants, updating in real-time. And ability to use model in lifecycle management. Basically, it means that there is no borders between the software, and they all are collaborated and models are not standalone.

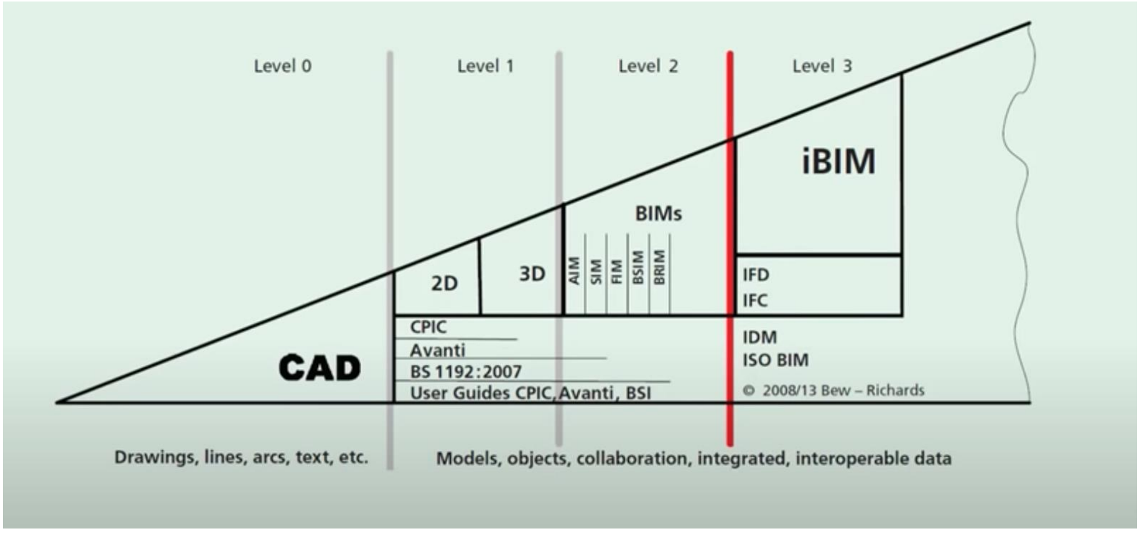


Figure 2. BIM Maturity model (The B1M)

Nowadays, most of companies who has BIM in their workflow, are on Level 1 or Level 2. Since Level 3 requires software that able to unite all workflow processes together, and also it requires engineers, who has understanding of whole BIM workflow process including knowledge about usage of other soft, where the export of model is planning to be done.

If we take a look at Autodesk Revit, being nowadays the most popular design software globally, it's noticeable that there is no above listed disadvantages. Last years since Revit 2017 version they develop structural part quite actively. Especially leaving open API for third-part developers, there is new addons coming for help to structural engineers.

Revit itself has tools for creating analytical model inside the physical one, that can be reviewed separately, adjusted, applied loads and exported. Reinforcement design performed very similar with Tekla workflow. And also Revit allows to work with topography, piping, and architecture, that could be a good basis for collaborated model, where all participants work in real-time in shared model.

The hope to the new possibility of design automatization for structural engineers comes with collaboration of Autodesk and SOFiSTiK AG addons on Revit - ”SOFiSTiK Analysis + Design”. This way, we can use Revit abilities of working with analytical model as pre-processor instead of SOFIPLUS for creating analytical model, applying loads, and generating FEM mesh with right material and section properties taking information from physical BIM model. We can avoid of doing double work creating every time the separate model for structural analysis and easily export it when we have any changes.

Additionally, also two more addons were considered:

Modelling process, and two-way connection with data of FEM analysis might be improved by "Reinforcement Generation" addon, that has tools for automatic reinforcement according the needed steel area taken from calculation results in elements.

And producing reinforcement detailing drawings from Revit model might be easier with "Reinforcement Generation" which has tools for drawing representation according to German traditions of drawings, that has quite much of similarities with Finnish Standards.

This way, we collaborate and connect processes of 3D modelling, FEM analysis and drawing producing using one Revit model, building up the single BIM workflow process for design, that might be used in all stages of project. Moreover, we leave a space to collaborate with other disciplines more easier since Revit might be used as a good platform for Architects and HVAC designers.

There is a need in team of engineers to develop good design of any construction object in a traditional way. Traditionally, each unit working separately at different aspects of projects. For structural part there is need at least minimum 2 people – structural engineer who is doing an actual design in FEM or analytically, and drafter/modeler who is responsible for drawing documentation or model. Or one person, who summary will spend the same amount of time as 2 listed above people because they only exchange by final results and performing independent tasks.

As in Henry's Ford idea of assembly line, each unit is doing a specific task and give the result to the next assembler. First – analytical model, then physical model, and then drawings. This system works, but there is one significant difference between fabric production assembly line and design of construction project – is changes. When changes are coming to the product which is on the middle of assembly line, there is a need to come it back to the beginning, and each assembler makes changes in that part that he was previously involved in. Even if change is quite small, it must be done in many places.

2.2 Research problem, research gap and research methods

Talking about available researches, there is article from Revit Structure blog by 10 Jan 2018, “An Overview of Structural Analysis for Revit”, reviewing the connection of analytic model in Revit and calculation process in Robot. Having absolutely identical process of using Revit as pre-processor for data input, any literature about connection Revit with any other FEM products might be used.

A vital part of speeches about connecting BIM and FEM into single workflow, were hosted by Russian department of PSS GRAITEC conferences.

Usually, the report or article says about what kind of workflow was used, but doesn’t open up all the solutions of handling limitations. The biggest valuable part of researches in that area is kept unpublished, because it has direct value for companies where research was made.

During the research, practical manual with detailed instructions for correct usage of Revit modelling tools for purposes of underground structures and tunnels was written for AFRY Finland Oy and kept for internal use.

Researches which are more focused on FEM part only with tests and comparisons of few elements types can be found in free access as master thesis works from different technical university students around the world.

Research methods include:

- Experiments and result analysis – different kinds of tasks were modelled and results compared between each other and other methods of calculations.
- Pilot project – application of new workflow on a real case example of maintenance tunnel.
- Surveys – finding information, recommendations, use-cases about following design method, BIM and FEM in particularity.
- Interviews with BIM managers and structural engineers about user experience, to predict possible limitations, wished expectations and most problematic issues in design.

2.3 Research objective and questions

Research objective is finding a way of correct modelling, to get the most realistic FEM model, and be able to make reinforcement design according to Finnish traditions of drawings representations.

Since Revit features about structural detailing is relatively new feature comparing with full history of this software development, and for structural engineer working with commercial and living buildings there is enough of instruments, families and templates, but for the infrastructure objects Revit UI and program logic meets following difficulties:

1.) Curved geometry and changing slopes. The Axis may curve in 2 directions, both having a radius curve by X global axis for example at plan view, and having curve by Y global axis at elevation view.
2.) Cross-sections of one element in infrastructure design may vary and don't be constant. A lot of custom elements must be used for cast-in-place concrete parts.
3.) Modelling components such as beams, slabs, and etc must be attached to work planes and certain levels. But in infrastructure design we don't make levels as a reference plan for placing elements, because it's often vary quite much from topography, and neighbour situation. Also most of elements must be placed from the plan view, but for infrastructure projects it usually not the most informative view and often facade view is more convenient for modelling. But in some cases, Revit allows to adjust some elements from the facade, after we locate it on the plan firstly.

So, why as an objective of research SOFiSTiK and Revit were taken, despite of its imperfections?
For now, there is no perfect tool for infrastructure design yet. They all has its own limitations and advantages. And as mentioned in previous chapter, it seems having its own ambitious and perspectives. So the final approach of research – to find out limitations for infrastructure design, the ways how it's possible to handle them, and avoid for making design of reinforcement concrete structures efficient, transparent, and easy-adjustable.

SOFiSTiK is powerful calculation complex that suit's the best way for design of unique and infrastructure objects because following features are realized there:

1.) Post-tension and prestressed structures
2.) Calculations by construction stages
3.) Composite cross sections

4.) Perfect with difficult geometry
5.) Automated report generation and summary of masses
6.) Moving Traffic loads according to Eurocode

So, the limitations in a chosen workflow are coming out from Autodesk product side as a pre-processor for analytical model. But at the same time, there is much of advantages, as just the possibility to easily export analytical model to SOFiSTiK is a very good feature that other BIM modelling solutions from Trimble or NEMETCHEK doesn't have at all.

2.4 Limitations and assumptions

Since the major focus of Autodesk Revit used to be on Architecture for long time, and later on Civil and Commercial buildings, there is few limitations of mechanics and logic of modelling.

Those are resulting the following assumptions:

1.) No possibility to make cut along curved line or axis.
2.) No possibility to make inclined levels.
3.) Curved walls cannot have modified profile if they have radial curve in plan, but might be adjusted by top and bottom to slabs (which might be inclined)
4.) Mass-modelling doesn’t create analytical modelling, but can be used as host for rebar.
5.) Not possible to define temperature inside and outside for defining temperature forces.
6.) Changing cross sections might be modelled by different elements with different cross sections, rigidly connected.
7.) Only 2 and 4 nodal elements might be modelled in Revit. Creation of solid FEM models is not possible.

3 FEM in theory

3.1 FEA as the main method of structural analysis

Before we start to create a model, we need to know logic of work FEM analysis software. We should understand the basic principles of Finite Element Method, mesh-generating, center-lines of beam elements, shell elements, and how connections should be modelled.

This questions comes from differences of modelling physical geometry of object and it's analytical model creation, being base for mesh generation. In structural design of reinforced concrete structures, we mainly use only beam and shell elements.

Membrane elements as well as truss elements are taking only normal forces (no bending), and it's possible to define from Revit as well.

Creating a solid model has sense for analyzing connections and edges of elements. For those purposes software IDEAStatica suits the best way, or high-end FEA complexes as NASTRAN, Abaqus, etc.

Spring and rigid links are used for boundary conditions, simulation of quantity compliance of connections and soil/bedding reaction simulations. That's also realized in Revit as well as defining degrees of freedom for them. Above listed most common types of finite elements used in civil engineering tasks are shown on Figure 3.

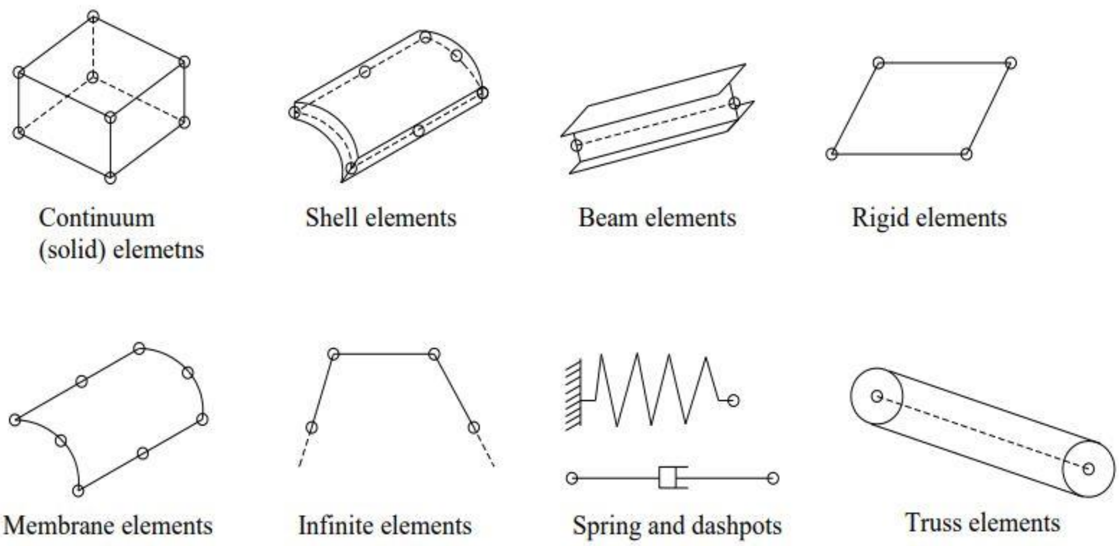


Figure 3: Most popular finite elements types used in civil engineering tasks (ABAQUS)

FEM nowadays used in computational analysis in many engineering disciplines, in structural design of structures, elements, connections. It's the one of the most widely used ways to analyze structure because of its flexibility to form, material, and structural type. It is possible to make simplified structure analyze with approximated properties as well as very detailed calculation of structure and it's connection with environment such as soil conditions, neighbor buildings, temperature and analyze composite structures.

Modern FEM analyze software allows to rich high level of automatization in calculations due to automatic mesh generation, library of materials and cross sections, verification according to local country's building codes. Besides pre-made properties and models of behavior, it's possible to overwrite it by custom values as well, manually codify conditions and mesh generation. This way, we get universal, adjustable easy-to use and powerful method of computational design to analyze stress, deflections and strain.

Example of generated FEM mesh model, exported from Revit is shown on Figure 4.

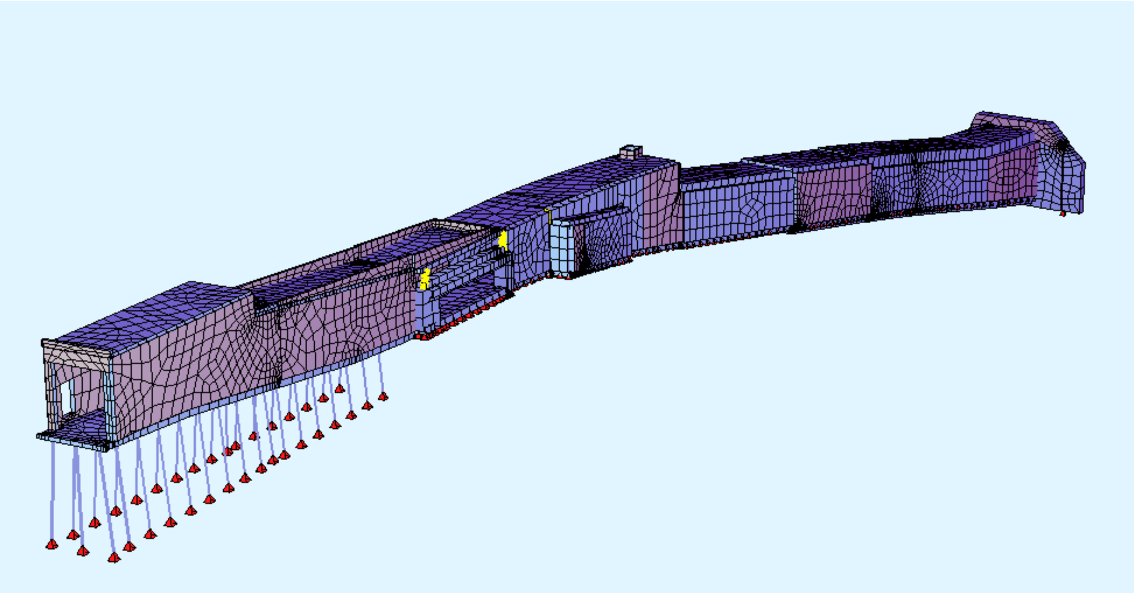


Figure 4. FEM model of tunnel, research objective

3.2 Displacement method as basement for FEM

Basement for understanding FEM might be classical displacement method shown on Figure 5. Main stage of solving tasks by displacement method is creating and solving system of canonical equations, where coefficients – are reactions in connections from single displacement in direction of those connections. In result, forming system of canonical equations relatively unknown nodal displacement.

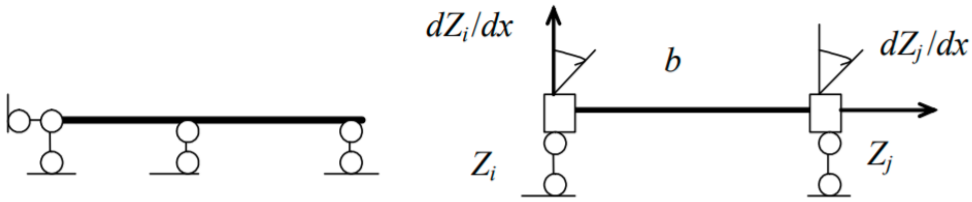


Figure 5: Displacement method for beam system. (Numerical methods for calculating building structures. Lebedev A.V. 2012)

The main idea of FEM consist of that any continuous magnitude in element might be approximated by discretized model, created by multiple bit-continues functions, defined in some number of sub-regions (elements). Usually those functions are polynomials – linear, quad, triangle and etc. Therefore, to find out values in element, we need to find values of magnitude in nodes of sub-elements.

So, difference with displacement method is that nodes might locate not only in connections of structure, but also in any place between the real nodes of element. Moreover, all nodes in bar connections and at supports are equal. Thus, all displacements are possible for considered calculation system. It allows more flexible take into account point loads, edge and boundary conditions.

Also, that's why visual control of displacement scheme according to engineering logic, allows to recognize correctness of calculation results.

3.3 Discretization and meshing of model

Main stages of creating discretized model for unknown magnitude:

1. In area under examination sets up number of points (nodes)
2. Values of continues magnitude are unknown and must be determined
3. Area of examination splits at number of sub-region (elements), having common points (nodes)
4. Continuous magnitude in each element approximate by each polynomial, which determined by nodal values.

Steps 1-3 are forming process called discretization.

Considering deflections *v* in cantilever beam example, continuous magnitude – function of deflection *v*(x). it's examination area – beam with length *l* shown on Figure 6.

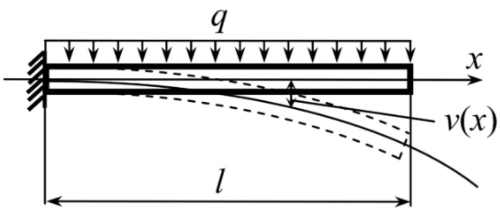


Figure 6. (Basics of FEM. Kondrat'eva L.E. 2007)

Set 5 points (nodes) shown on Figure 7. Determines values of deflection at each node: v_1, v_2, v_3, v_4, v_5.

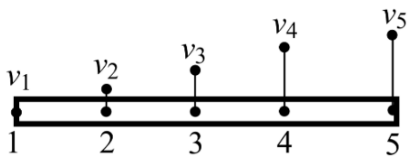


Figure 7. (Basics of FEM. Kondrat'eva L.E. 2007)

Approximated function – linear by x-axis polynomial, because each element has 2 nodes. Final approximation *v(x)* – four bit-continuous functions, each one determined on separate element shown on Figure 8. Unknown nodal values *v(x)* must be corrected this way, to be approximated the most closest way to real function *v(x).*

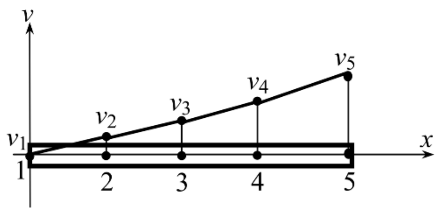


Figure 8. (Basics of FEM. Kondrat'eva L.E. 2007)

If unknown continuous magnitude determined in 2D or 3D area, approximated function exist by x and y, or x, y, z axis coordinates. 2D area usually determined by triangle (with 3 common nodes) or quad elements (with 4 common nodes). And 3D object usually determined by tetrahedron (usually for soil mechanics simulations) or parallelepiped (for solid element analyze). Approximated functions are shown in that way as a surface.

Triangle areas are the simplest to make discretization by choosing *n* number of nodes from each side. Then number of triangle elements will be $(n-1)^2$
Quad areas are usually discretized by connecting nodes from opposite sides. If number of nodes on opposite sides is equal to *n* and *m* for opposite pair sides, then number of quad elements will be equal to formula $2(n-1)(m-1)$.
Both triangular and quad meshes are shown on Figure 9.

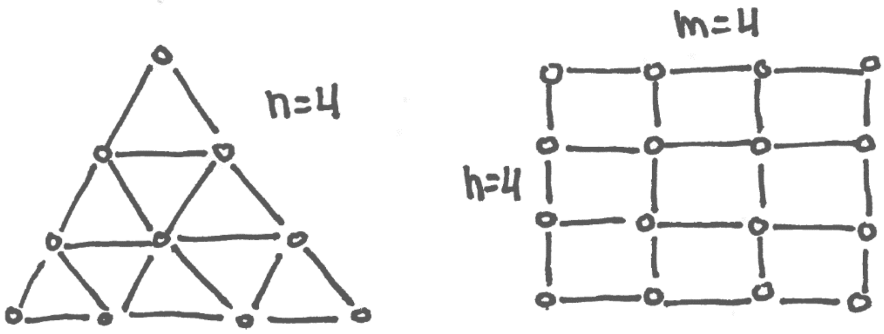


Figure 9: Triangle and quad element meshing by equal elements.

But it's quite rare when examination area should be split by equal size of elements. Nodes might be placed on element in following points:

1. Where changes of material, it's properties are occurring.
2. Where geometric characteristics of cross section is changing.
3. Where point loads are applied
4. Where distributed area load is starting/finishing
5. Where boundary conditions are changing

Few examples of that meshing is represented on Figure 10.

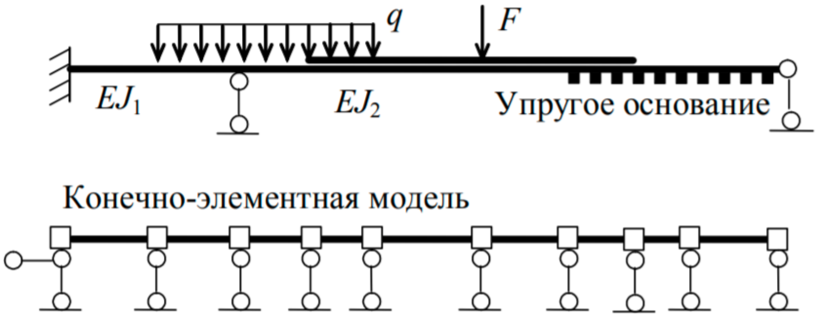


Figure 10: Possible FEM model of beam system (Numerical methods for calculating building structures. Lebedev A.V. 2012)

Discretization might have bigger points density in parts where bigger difference of results is expected for magnitude (Stresses), for example in continuous beam over support, or at slab edges, around openings.

3.4 Main parameters of finite elements

DoF – Degrees of freedom – one of the main parameters, number of all possible movements in nodes of finite element. Number of DoF defined by chosen structural type, and meaning of solving task. Such as free movement by x,y,z -axis and rotations around z,y,z – axis, forming 6DoF.

Depending of quantity DOFs in element's node forming matrix of rigidity of finite element. But in FEM softwares this systems are kept invisible and no need for an engineer to change it manually and it's enough to operate by DOF only by setting up boundary conditions representing the behavior of structure for analytical elements. Finite elements inside one bar or shell has rigid nodal connections between each other.

Dimensional in space – might be as 1D (bar), 2D (shell), 3D (solid).

System of approximating functions – allowing to determine deflections in any point of element through its degrees of freedom

Physical behavior – determining dependence between inner stresses and deflections defined by material behavior theory and physical properties.

Number of element – number of elements in in exanimated area formed my matrix of rigidity. If difference between numbers of elements in one area is bigger, it will case longer time of calculation caused by more complicated matrix with wider raw.

Combination of loads and actions that might be taken by element – might be normal forces only (membrane elements), bending only, or both.

Shape – usually it's simplest geometrical shapes from line to tetrahedron. Represented on Figure 11 and Figure 12.

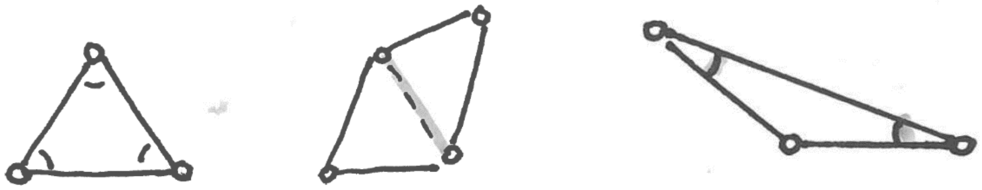


Figure 11: Triangle element shapes

Perfect shape of triangle element giving the most accurate calculation results - equilateral triangle. Recommended minimum angle $a = 15^{o}$

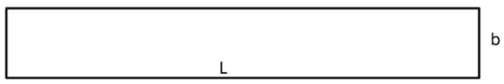


Figure 12: Quad element

For quad element recommended ratio l/b < 10

If shape of elements usually created by automatic meshing, so there is might be degenerate elements such as shown on Figure 13:

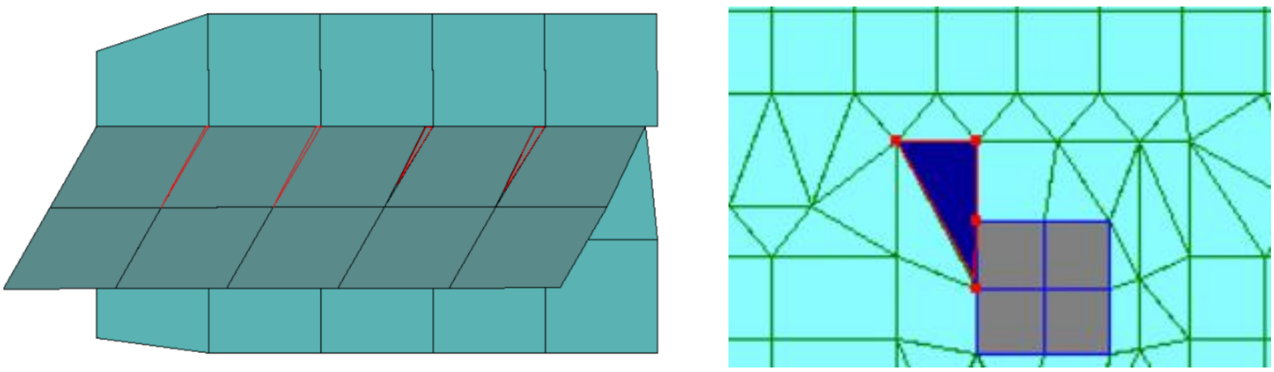


Figure 13: Degenerate elements. (Lira database)

1. Triangle element with sharp angle (less than 15^{o})
2. Quad element with very obtuse angle (almost 180^{o})

We should always try to avoid appearance of those elements and correct mesh if possible because it might cause of increased stresses (and then reinforcement area during design operations)

3.5 TRI vs QUAD elements

Nowadays many of modern FEM softwares are able to make automatic meshing, and it's possible to choose if mesh must be performed by Triangle elements (TRI) or Quadrilateral (QUAD) just by choosing it by one click in settings.

According to research of EnterFEA source, calculation results of same object was compared with triangle and quad mesh. Meshing was done that way, so each quad element was divided in half by crossing a line between 2 closest nodes in quad element. So they got 2 times more elements in TRI model than in QUAD model. And it took similarly the same time to compute. But results were quite different. Diagram of ratio displacement to load is shown on Figure 14.

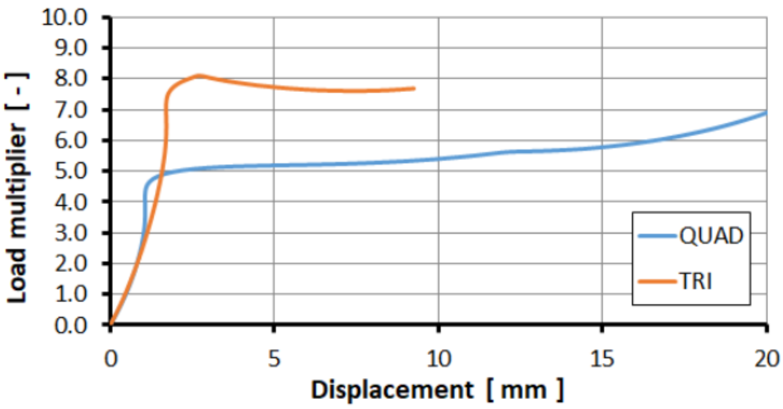


Figure 14. Displacement to load diagram (EnterFEA)

The difference is that, that TRI model has shown extra stiffness, it came to the realistic stress value only when load was multiplied by 8. Compared with QUAD, that required multiply by 5 to rich the same stress. Comparison analysis is shown in Table 1.

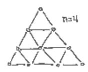
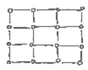
	TRI 	QUAD 
Meshing difficulty	Faster and easier to mesh	More difficult to mesh
Time of computing	longer to compute, because we need to create more of them	Faster to compute
Result accuracy	60% overprediction of capacity even with double number of elements	Result is relatively accurate even with bigger mesh.

Table 1. Comparison of TRI and QUAD

Triangle models shows the right result for deflections, but to define correct stresses, we might need to create just too much of elements that would overload model. Therefore, since in design of reinforcement structures we are most interested in stresses, and meshing takes much less percentage of working process than computing, it's better to use QUAD elements always for reinforcement design.

3.6 Nodal and Element values

SOFiSTiK allows to choose what values take into account for reinforcement design – nodal or element values, so It is important to understand the nature of those values.

Each quad element has 4 shared nodes, that are common also for neighbor elements. This means that one single node can have several canonical equations because it belongs to few elements.

The way, how stiffness matrix works, is that it creates shape function for each element, and put it into matrix separately for each element. With assuming that deflections in every node must be that same, it makes the assembly of global stiffness matrix.

That means that solver firstly get deformation in nodes, and then using shape functions we can get strain calculated in element. And knowing strain, by stress-strain relation we can get stresses in element.

But because in each common node we get few strain values, for each element that it belongs to, the FEA postprocessors will show averaged value in nodes. This example is shown on Figure 15 from EnterFEA article.

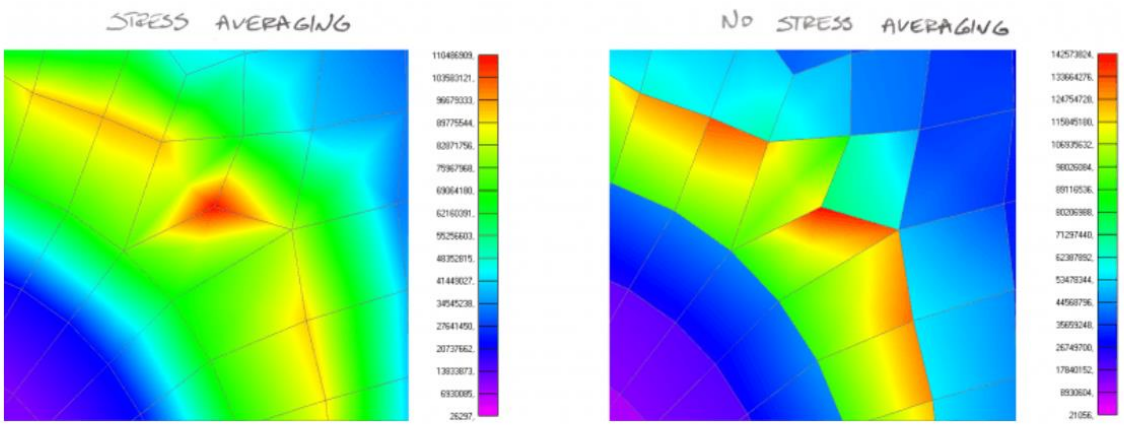


Figure 15. Averaged and not averaged stresses (EnterFEA)

Again, to reduce difference between values in a common node from different elements, the solutions is mesh refinement to get nodes closer to each other. But also good quality of element shape is important.

4 Material behaviour in SOFiSTiK

4.1 Baumann

Following design method is based on Baumann theory (Der Bauingenieur 47 1972, pages 367-377) and it's similar with sandwich model according to EN 1992-2 described in Annex LL:

For the analysis of reinforced concrete elements that can be cracked, the usual plate element with 3 normal forces, 3 bending moments and 2 shear forces is not enough and sandwich model should be used.

In sandwich model, two outer layers resist the membrane forces, and the inner layer carries the shear force and represented on Figure 16. Considered forces in model are shown at Figure 17. And Baumann model is shown at Figure 18.

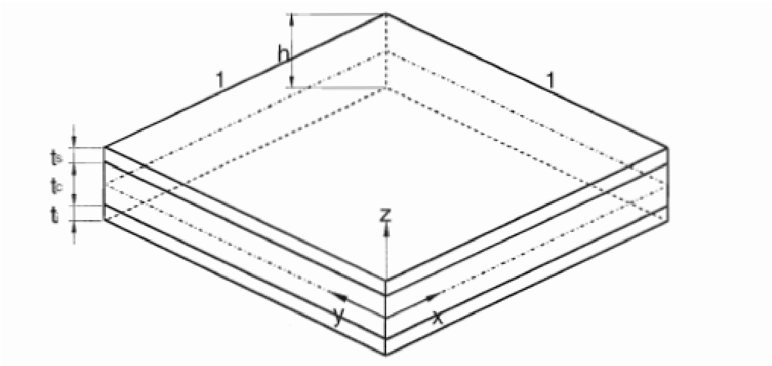


Figure 16. The Sandwich model (EN 1992-2)

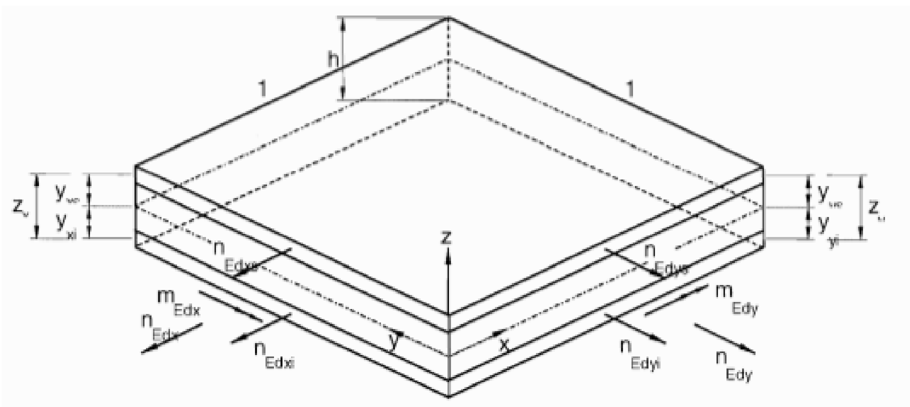


Figure 17. Axial actions and bending moments (EN 1992-2)

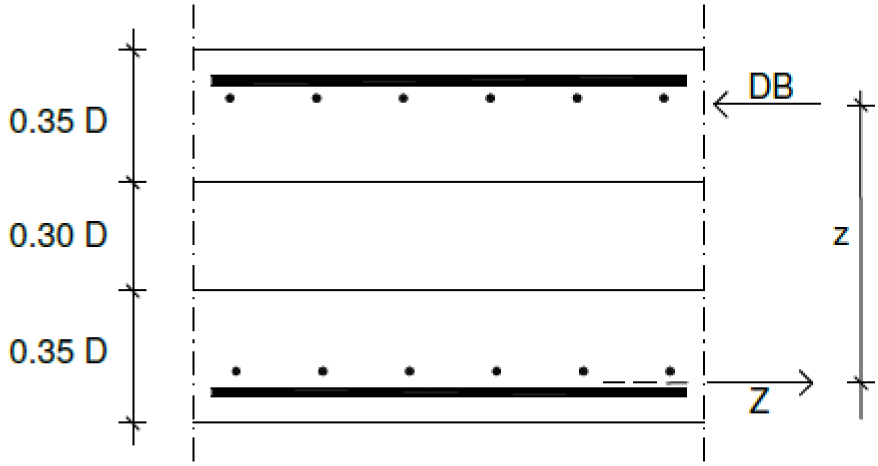


Figure 18. Fictitious disks and lever arm in the Baumann design (SOFiSTiK AG)

Baumann model allows to take compressive and tensile forces and consider them for each disc separately with pre-calculated lever arm. For each reinforcement layer, the lever arm is calculated separately. Fictive disc forces foe each considered shell side caclulated according to formulas 4.2, 4.2 and 4.3.

$$Ax = \frac{Nx}{2} + \frac{Mx}{z} \quad (4.1)$$

$$Ay = \frac{Ny}{2} + \frac{My}{z} \quad (4.2)$$

$$Axy = \frac{Nxy}{2} + \frac{Mxy}{z} \quad (4.3)$$

The whole algorithm of calculating according to Baumann is described by SOFiSTiK AG in BEMESS manual 2020, p. 6-7.

4.2 Layer design

The previous method is facing a problem that usually the main tension direction in slabs is different from direction of reinforcement. Thus, we are getting the rebar mesh deformed in a rhombus shape, that need a compression strut (that we don’t have in reality). Example representing the tension in not direction of reinforcement mesh is shown on Figure 19.

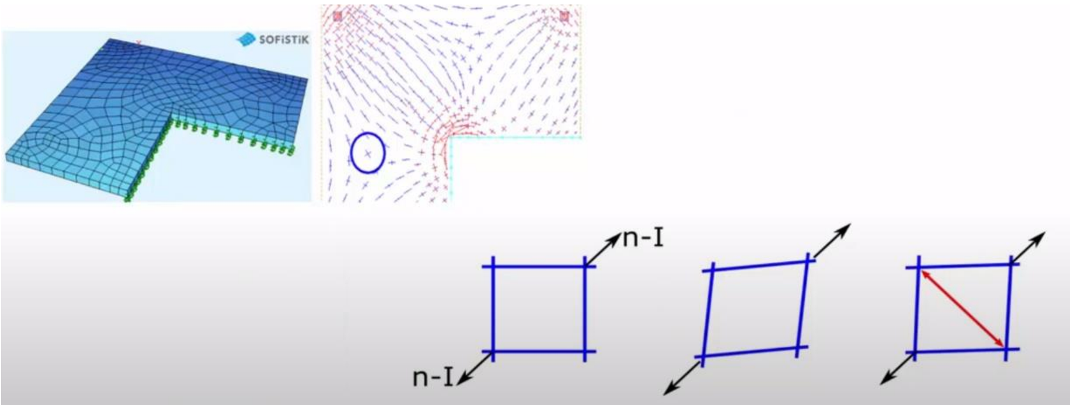


Figure 19. (SOFiSTiK AG)

To predict realistic behavior of material, method was presented by SOFiSTiK AG in 2017 at München Concrete Seminar and implemented in version 2018. For exact iteration of the strain state 6 external forces are taken into an equilibrium with 6 inner forces calculating 3 strains and 3 curvatures.

This way, nonlinear work law for concrete and steel is taken into account, and recommended to use for SLS calculations. Stresses in material withing using this method are shown in Figure 20.

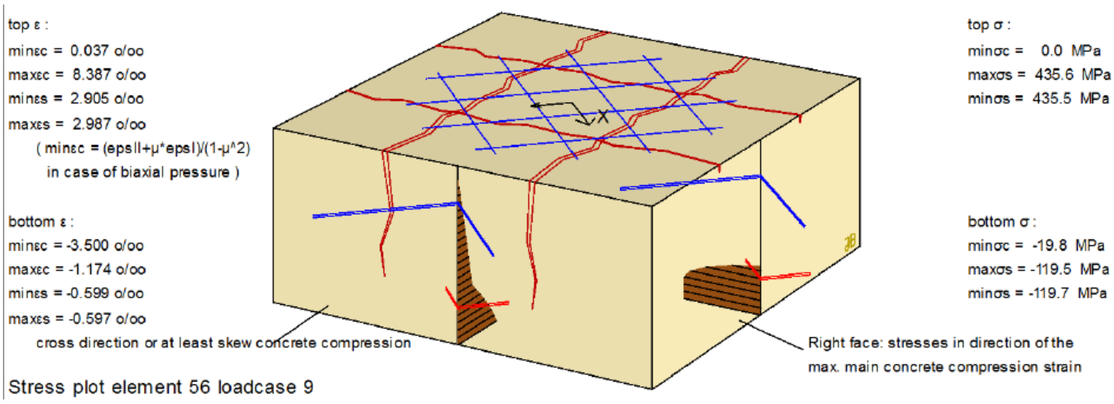


Figure 20. Real stress distribution with layer design. (SOFiSTiK AG)

Algorithms of calculation inner and outer forces for iteration of strain state is shown on figure 21.

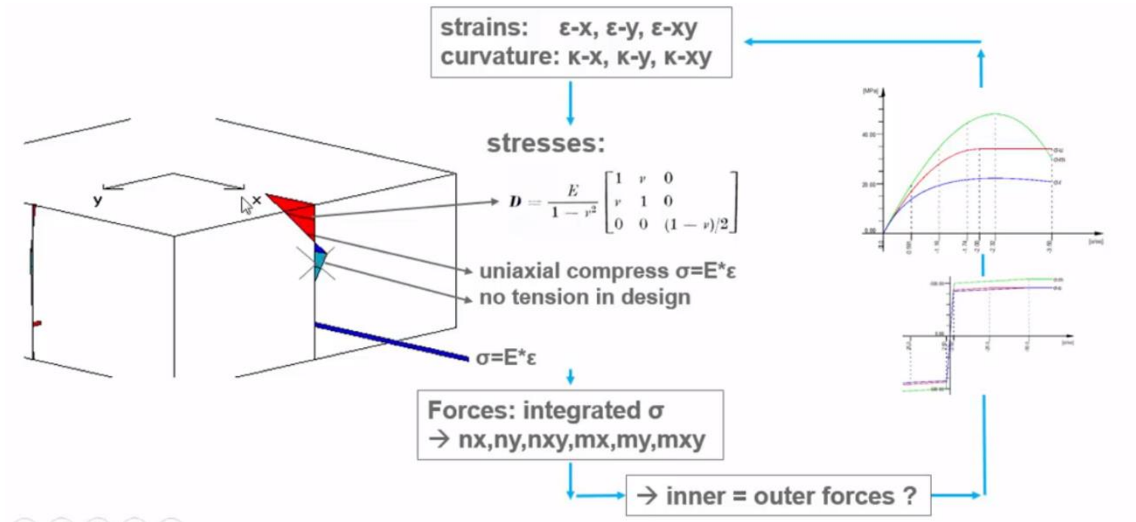


Figure 21. (SOFiSTiK AG)

Accordingly to the name of a method, the main idea consist of dividing layer on few ones, and each would be calculated separately. As it's shown on the Figure 21 - the first approach is at defining strains, and then we get stresses. In case of biaxial compression, Poission's ratio is taken into consideration. In case of uniaxial compression and tension is on the other side, we make an uniaxial behavior (4.4).

$$\sigma = E \cdot \varepsilon \qquad (4.4)$$

After that, if inner forces and outer forces are not in equilibrium, we correct our curvature till we get them in equilibrium.

5 Practical examples

5.1 Overview of Revit UI possibilities for pre-processing

5.1.1 Analytical model

Since the final outcome of our pre-processor is model consisted of bars, shells, links and springs, that looks slightly different from a physical model and has absolutely different logic of model creation, that’s very important to follow the right logic of modelling for correctness of model behavior.

Slab are modelled by shell element when thickness to span ratio less than 1/10. So maximum 1m thick slab might be modelled by shell element if span is 10m for correct behavior. Example shown on Figure 22.

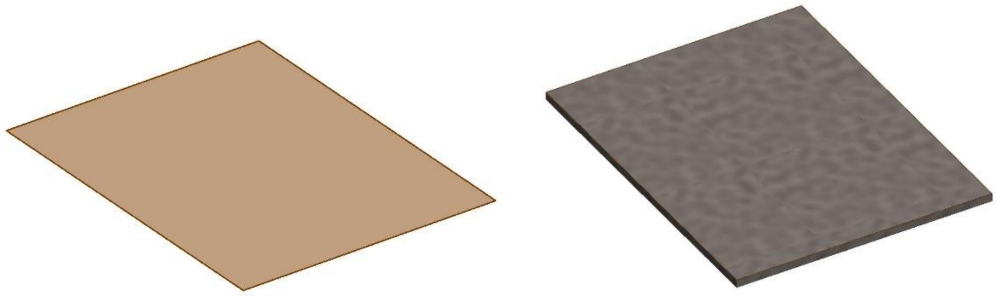
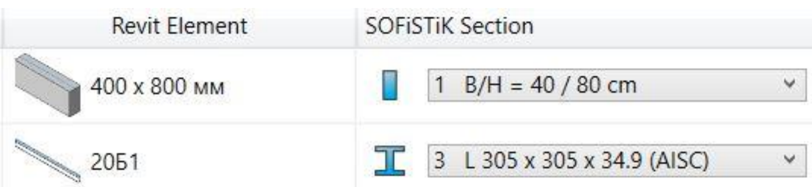


Figure 22: Analytical and physical slab element

Beam might be modelled by bar element if ratio height of cross section to length of the beam less than 1/10. So maximum 1m height beam might be modelled by bar element if span is 10m. Example shown on Figure 23.

Figure 23: Analytical and physical beam element

We must always set-up cross sections to bar elements (2-nodal) and thickness to shell elements (4-nodal). Section mapping between Revit model and SOFiSTiK database is shown on Figure 24.



Revit Element	SOFiSTiK Section
400 x 800 мм	1 B/H = 40 / 80 cm
20Б1	3 L 305 x 305 x 34.9 (AISC)

Figure 24: Section mapping in Revit

In Revit analytical bars might be adjusted thorough ***"Analyze"-tab > "Analytical model tools"-panel > "Adjust"*** command.

If there is a need to connect bars by link connection, then the tool ***"Analytical Link"*** creates rigid connection between 2 nodes. It's also possible to define DoF of the node by free/fixed release for each direction or writing spring characteristic.
In case when we need to connect bar to a place where is no node, (for example in the span of the beam or on the slab) we need to create node element by shape editing or by splitting the element. View of rigid link in Revit is shown on Figure 25 and 26.

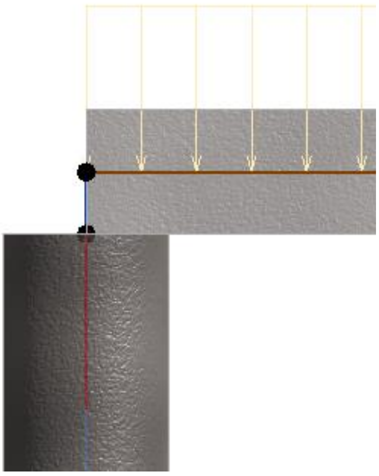


Figure 25. Analytical link in Revit

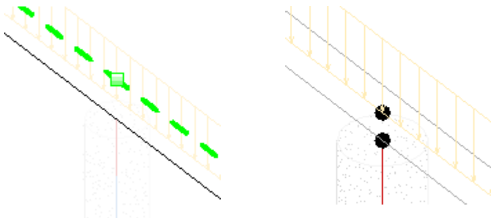


Figure 26. Analytical link in Revit

Another way to create analytical link – might be done in **Properties Menu** of selected analytical bar. Menu is shown on Figure 27. There is a tick for creating analytical link to the closest object.

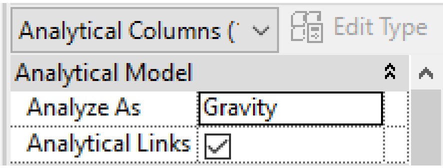


Figure 27. Properties of Analytical Element in Revit.

Not all physical elements are having analytical model. Only the following ones are able to create analytical model:

- Structural Column,
- Wall: structural,
- Floor slab: structural,
- Beam,
- Structural Foundation: Isolated,
- Structural Foundation: Wall,
- Structural Foundation: Slab

If we are assuming to do our model only by those tools, that are usually enough for solving practical engineering tasks, then automatically created analytical model inside the element recognizes physical properties, cross sections for bar analytical elements, shape and thickness for shell analytical elements.

The big advantage is that custom cross-sections are recognized by SOFiSTiK, and reading the geometry from Revit model, a new cross section is automatically created in SOFiSTiK database.

Some solvable difficulties are coming when we try to use IFC model for structural analysis. Since IFC is open format for sharing data with OpenBIM approach, structural analyse is not the main deliverable there. This is caused because different FEM programmes are using different algorithm of making FEM models, has different pre-processors and some of them are not creating physically visualised model at all. So, unlike of meshing 3D geometry, transfer of analytic model through OpenBIM concept is not realized yet.

The special feature of SOFiSTiK plugin allows to create analytical model as in IFC imported models, as well in Revit objects that doesn't contain analytical model. The workflow build that way, of creating ***"Model In-Place"*** model in categories that are able to create analytical model. By choosing subcategory ***"Analytical model"*** it is possible to sketch boundaries where a new analytical shell will be created. Subcategory menu is shown on Figure 28.

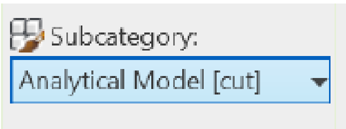


Figure 28. Subcategory from plug-in

Having an analytical shell modelled, without difference was it modelled by native Revit tools, or from plug-in - it is possible to define structural properties, edge releases, thicknesses and stiffness by ***“SOFiSTiK: Structural properties”*** menu shown on Figure 29.

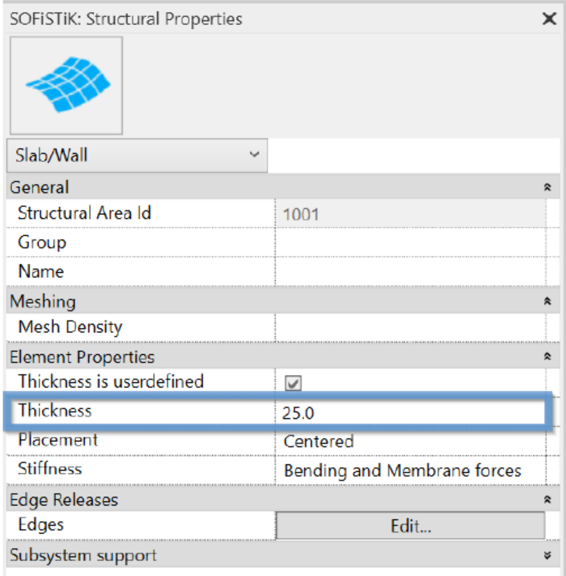


Figure 29. Interface of menu from plug-in

Location of analytical bar or shell inside the element might be adjusted by two ways:

1. From Properties menu shown on Figure 30, by Analytical Alignment to operate projection. This way is recommended by developer since the alignment is the most accurate way for placement, and it continue to behave the same way when changes in location of element are occurring.

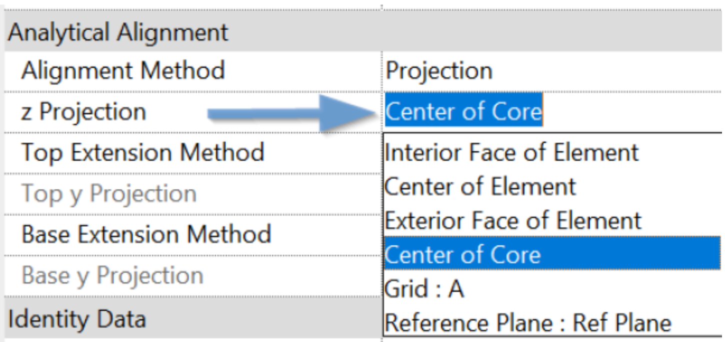


Figure 30. Properties menu of analytical model

2. Manual editing of analytical shell/bar shown on Figure 31 and Figure 32. The advantage of this method is free control of placement by possibility to drag, snap boundary points in 3D view. But behaviour and flexibility of transforming analytical models is vary. For example, beam/columns bars can freely move in 3D space and drag to any elements. Shell models made by Slab element can freely transofrm and snap contour to nodes and other analytical elements in the

work plane that physical model was scketched or aligned. And shell models made by Wall element can only be changed by changing profile of wall, analytical adjustment described above, or Wall-adjustment tool, that only allows automatically connect edges of selected walls.

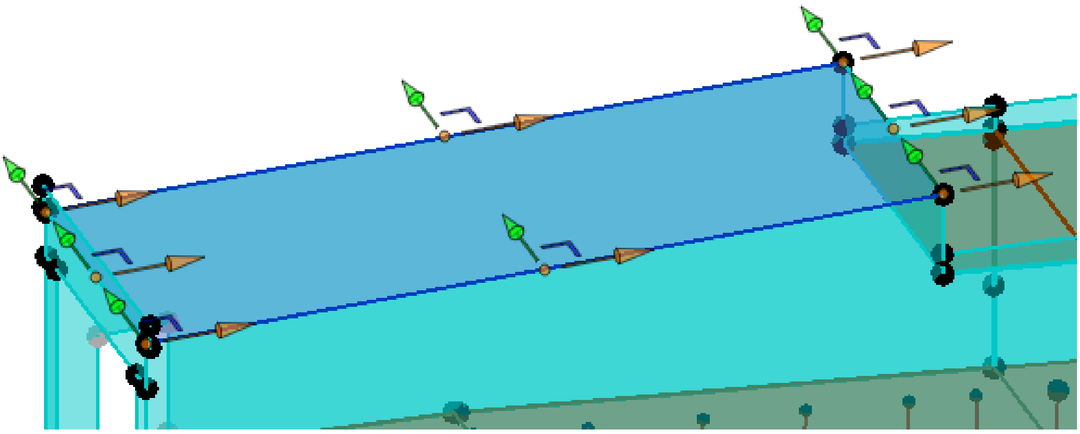


Figure 31. Manual adjustment of model generated from Slab element.

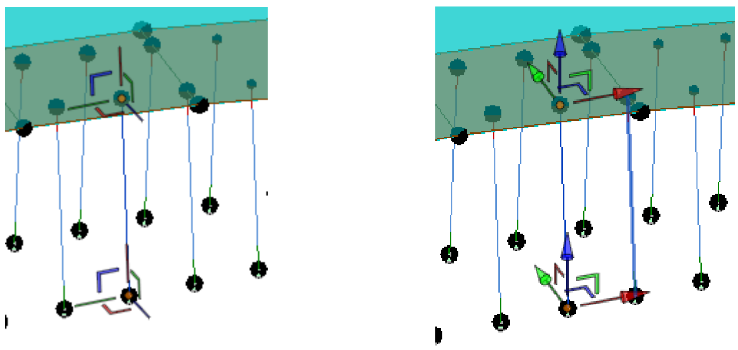


Figure 32. Local and global coordinate drag marks.

5.1.2 Boundary conditions

Boundary conditions might be attached to points, along lines, or by surfaces. Native Revit families has three pre-sets such as pinned, roller, and rigid. In model they are represented as it shown in Figure 33.

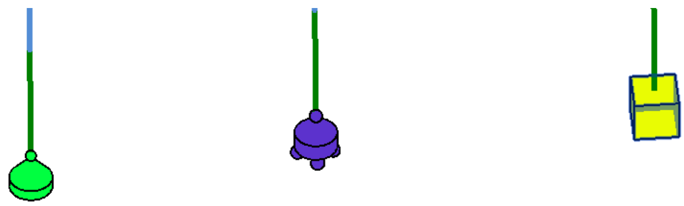


Figure 33. Boundary conditions

But there is possibility to create custom boundary conditions by defining DoFs manually, as well as defining rigidity of node by spring modulus in $\frac{kN}{m}$ for linear translation or $\frac{kNm}{\circ}$ for angular rotation that must be defined from by known displacement for applied force. It can be filled in properties menu which shown on Figure 34.

From the simple Hook's Law formula (5.1), there is interpretation for material resistance by Edme Mariotte (5.2).

$$F = k \cdot x \qquad (5.1)$$

Where F – Force

k – spring modulus

x – displacement

$$\partial_z = E \cdot \varepsilon_z \qquad (5.2)$$

Where ∂_z – Stress,

E – Young modulus

ε_z – deflection

Translation in	
X Translation	Spring
X Spring Modulus	1.00 kN/m
Y Translation	Fixed
Z Translation	Fixed
Rotation about	
X Rotation	Spring
X Spring Modulus	1.0 kN-m/°
Y Rotation	Release
Z Rotation	Release

Figure 34. DoF's and rigidity defining in Revit Properties menu for boundary condition.

In case of simulating soil mechanics, that parameter represented by coefficient of subgrade resistance C (5.3) and should be taken as table value for known soil type.

$$C = \frac{P}{S} \qquad (5.3)$$

Where C – Coefficient of subgrade reaction (bedding value)

P – Pressure on the ground area

S – Settlement (displacement)

5.1.3 Loadings and loadcase definition

Revit interface also allows to apply loads and define loadcases with load natures. Interface for applying loads is shown on Figure 35. Not only evenly distributed loads are possible to define in Revit, but also linearly uneven distributed. It performs by defining point at area of loading, and writing load values in those 2 or 3 points. Revit will linearly interpolate and extrapolate result by following are according to input values and point locations.

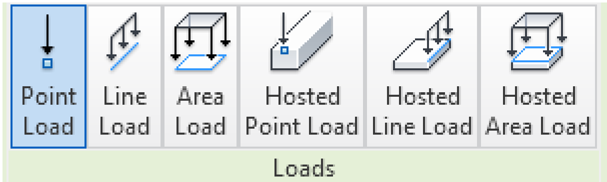


Figure 35. Possible ways of applying loads

Loadcase definition menu is shows on Figure 36 and mapping with SOFiSTiK load actions show on Figure 37.

Load Cases

	Name	Case Number	Nature	Category
1	Pysyvät (DL)	1	Dead	Dead Loads
2	Hyötö (LL)	2	Live	Roof Live Loads
3	KK1	3	Transport	Live Loads
4	KK2	4	Transport	Live Loads
5	KK3	5	Transport	Live Loads
6	Törmäys poikki	6	Accidental cross	Accidental Loads
7	Törmäys pitki	7	Accidental longitudunal	Accidental Loads

Load Natures

	Name
1	Dead
2	Live
3	Transport
4	Accidental cross
5	Accidental longitudunal

Figure 36. Load Cases in Revit

Revit Document	Revit Load Nature	SOFiSTiK Action
Lohko_1_2	Dead	G dead load
Lohko_1_2	Live	Q variable load
Lohko_1_2	Transport	Q variable load
Lohko_1_2	Accidental cross	A accidental loading
Lohko_1_2	Accidental longitudunal	A accidental loading

Figure 37. Load actions mapping

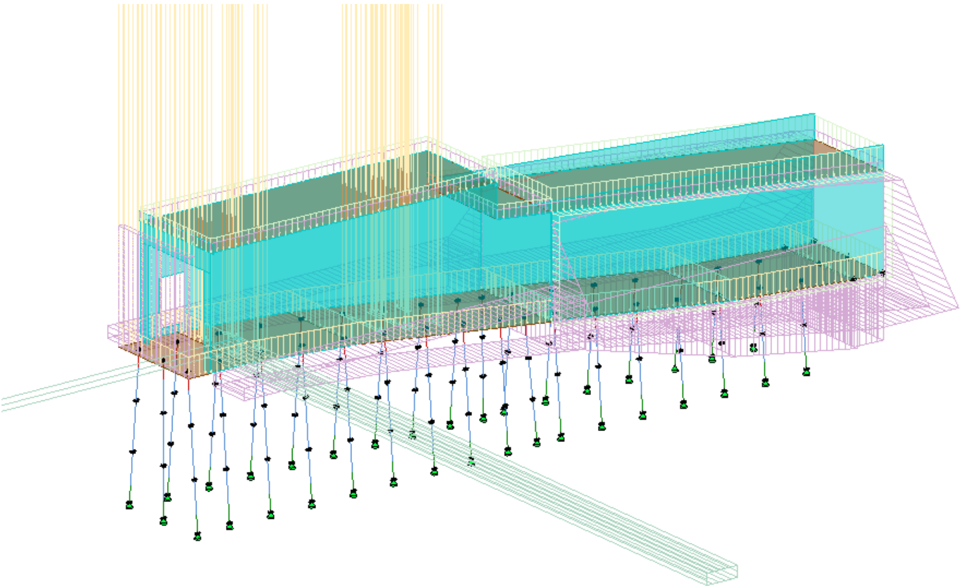


Figure 38. Analytical model with soil simulations and applied loadings

5.1.4 Material mapping

As well like for Cross section and Load mapping, there is as well material mapping between Revit Library and SOFiSTiK database. UI of menu is shown on Figure 39, where it is also possible to adjust material properties, define behaviour and bedding.

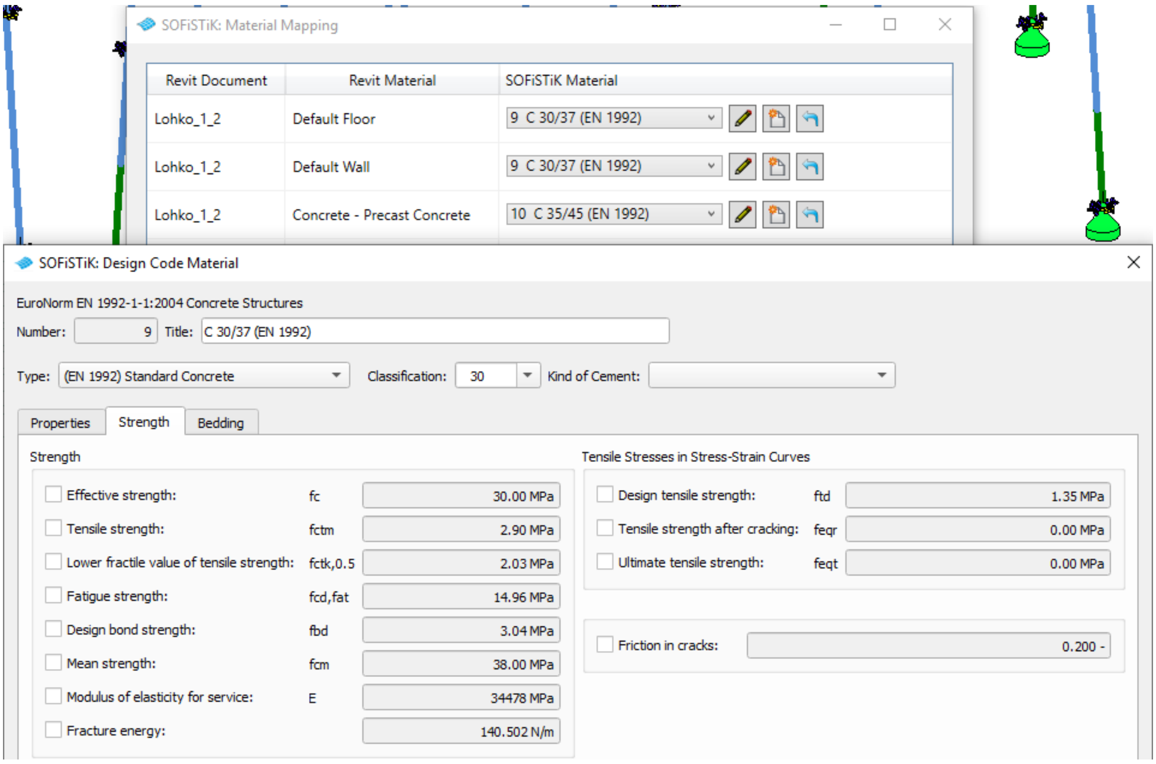


Figure 39. Material mapping and properties from Revit Interface

5.2 Bar placing in beam/column elements

5.2.1 Possible options

Revit allows to adjust manually the analytical bar placement inside beams, columns and slab elements. Depending of our physical aim of task, we have following options to model analytical model connections shown on Figures 40-45:

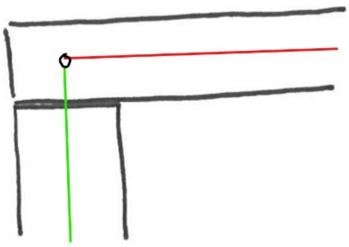


Figure 40: Adjust to the centerline

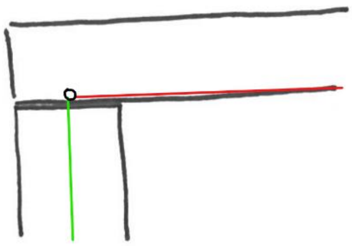


Figure 41: Adjust to the bottom

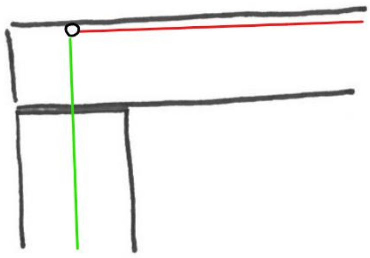


Figure 42: Adjust to the top

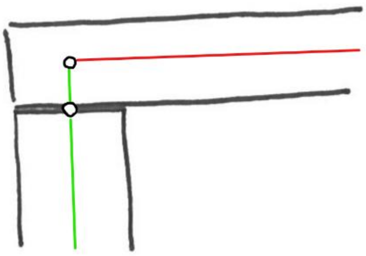


Figure 43: Insert rigid connection

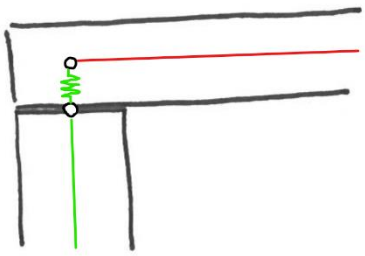


Figure 44: Insert spring connection

Solution that would gave the most accurate calculation result would be to insert spring connection with rigidity and properties of concrete.

Practically just rigid connection is often might be made. Being attached fixed to beam, and pinned to column. It has a sense to model when ends of bars are relatively far. For example, example in composite steel bridges when beam has a big height

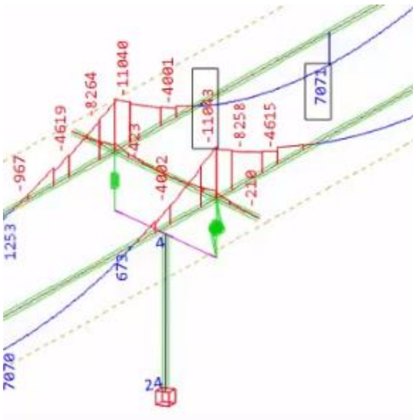


Figure 45: Bridge support connection

5.2.2 Effective length - Eurocode paragraph overview

In column and wall elements, the analytic bar must be always in the center of area of support, because according to the EN 1992-1-1, chapter 5.3.2.2, the effective span, l_{eff} (5.4) of a member should be calculated as follows:

$$l_{eff} = l_n + a_1 + a_2 \qquad (5.4)$$

Where l_n - the clear distance between the faces of the supports; values for a_1 and a_2 , at each end of the span, may be determined from the appropriate a_i values in Figure 46 from Eurocode where t is the width of the supporting element as shown.

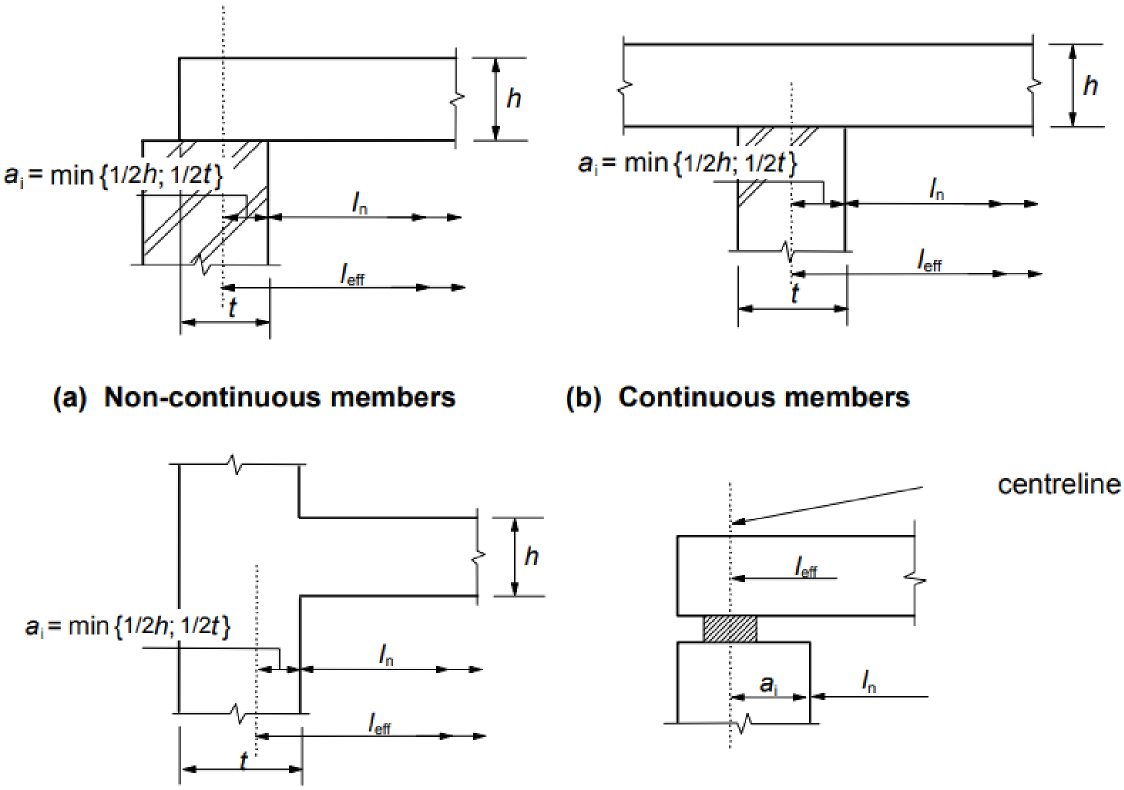


Figure 46. (EN 1992-1-1)

In all cases if there is no special support plate, and beam/slab fully cover the wall/column, the a_1 and a_2 are equal to half of the wall/column width.

In cast-in-situ concrete structures that is usually the case that should be used representing real structure behavior.

5.2.3 Rigidity of nodes for precast elements

In concrete structures with precast elements, the rigidity of connections must be taken in account due to Normal Force, Bending and Twisting Moment shown on Figure 47.

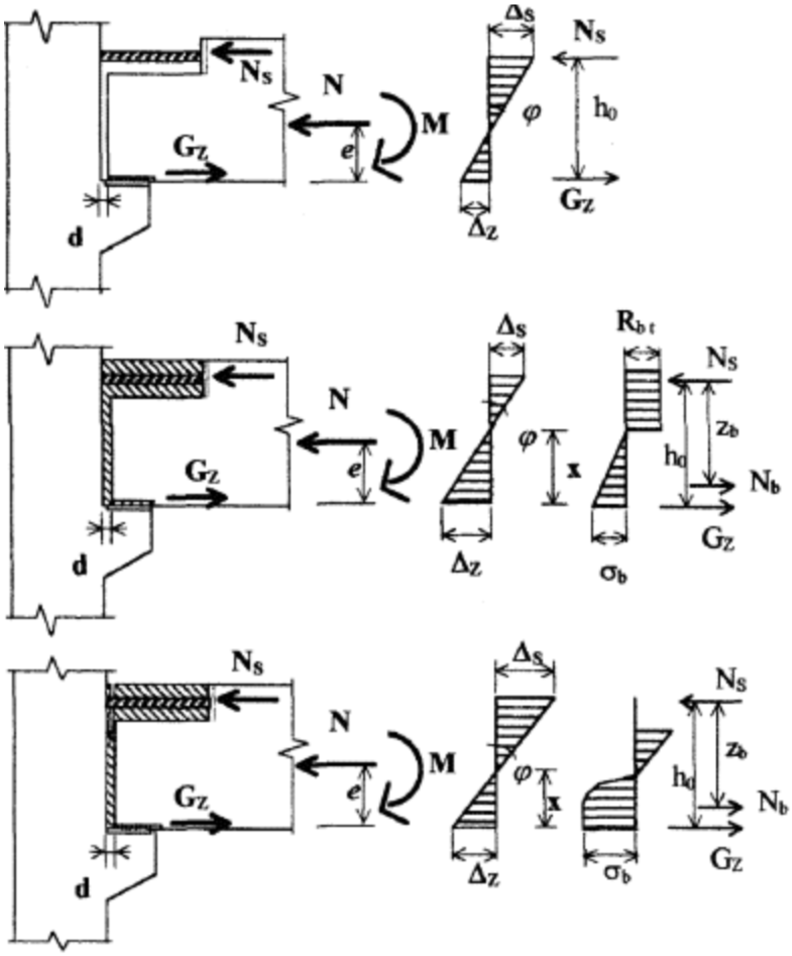


Figure 47: scheme of changing strains in column-beam frame connection

1. – construction stage

2. – casted seal, before cracking in tension zone

3. – stage after crack appearance

(Recommendation for calculating frame of multi-story buildings with rigidity of nodes precast concrete elements. Moscow 2002)

Criteria of ULS of nodal connection should be maximum allowed rotation angle, which determines by yield stress in rebar, concrete compression or shear of cast-in fixings.

Nodal rigidity coefficient determines by formula (5.5) and illustrated by Figure 48:

$$C_{\varphi} = \frac{M}{\varphi} \qquad (5.5)$$

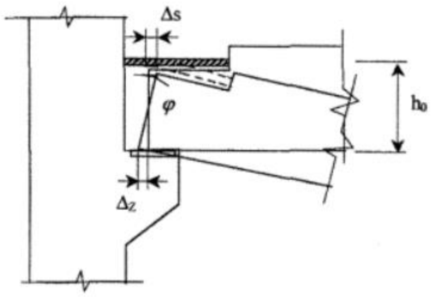


Figure 48: $\varphi - degree\, of\, rotation$ relatively column
(Recommendation for calculating frame of multi-story buildings with rigidity of nodes precast concrete elements. Moscow 2002)

But this case is rarely applied in tunnel and underground design since most of all connections are cast-in situ connected by overlapping bar layers both on compression and tension zones that makes connection absolutely rigidly behaving and transferring all loads without rotation in connection.

5.2.4 Test in SOFiSTiK

To check if nodal connection links effects on the Moment Peak Smoothing in slab, there is was made an experiment:

Two models were made with absolutely identical structure type, materials, cross sections and thicknesses. The only difference was in connection of column to continues slab. In the first model it is performed by direct connection, with increased calculated length of column according to Figure 40/42 (Chapter 5.2.1). In the second model rigid links was inserted to aim real calculated length of column according to Figure 43 (Chapter 5.2.1).

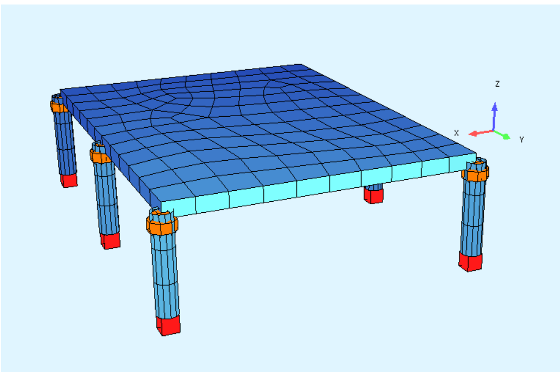


Figure 49: Without links. Column bar attached to the slab

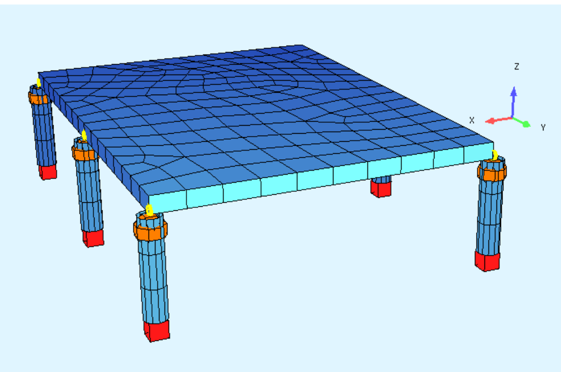


Figure 50: With connection link. Column bar has real length.

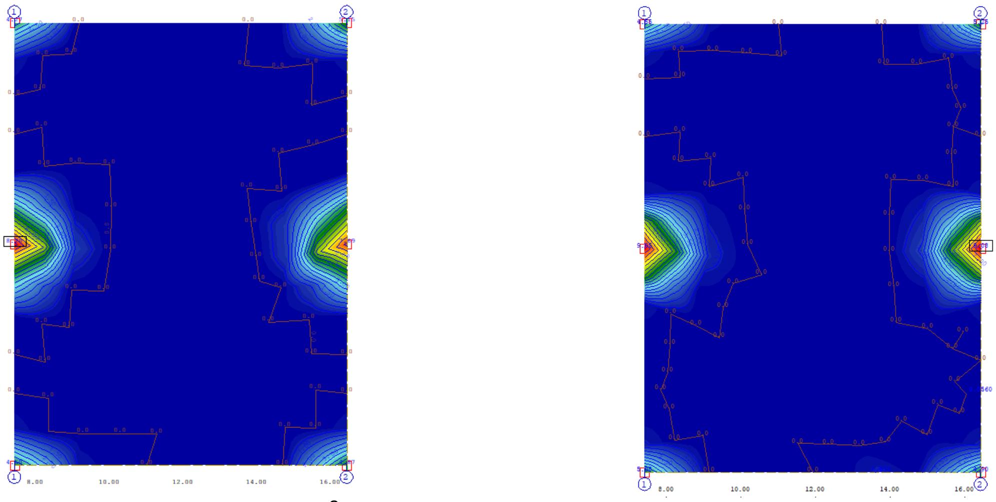


Figure 51: Area [mm^2] longitudinal top reinforcement by Y direction

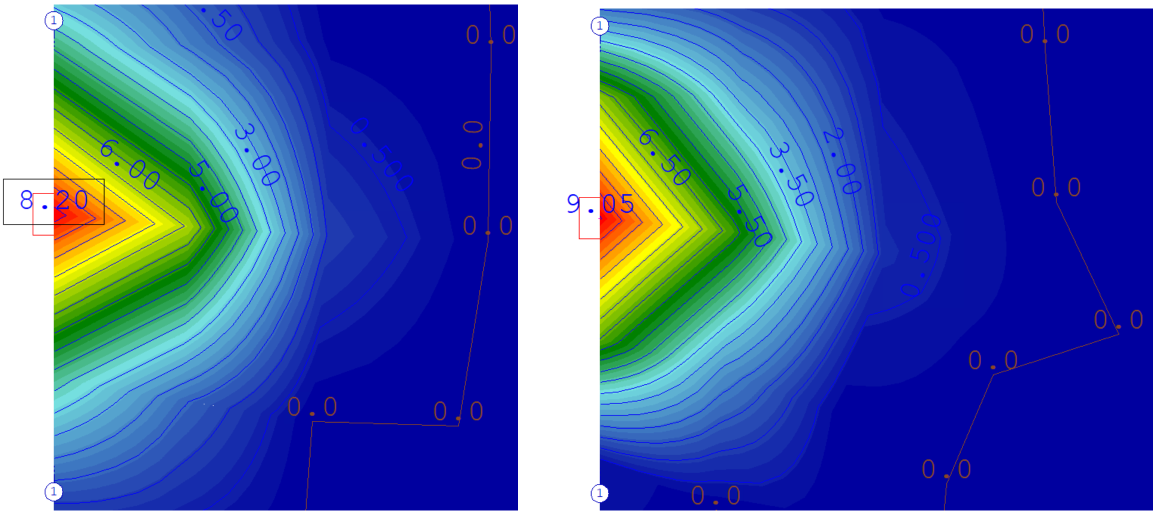


Figure 52: Area [mm^2] longitudinal top reinforcement by Y direction over central support

As we can see from postprocessors diagrams of reinforcement on Figure 51 and 52, it's nearly the same reinforcement area in two tested models. The difference around 10% in maximum results might be caused because of different meshing, that is actually influence more on values.

Peak smoothing is working in both cases, because reinforcement decreases if we set up wider column.

5.3 T-beams, TT-slabs

5.3.1 T-beam phylosophy

Typical bridge spans over the highways is often build using prefabricated T-beams, casted together on site and working as slab with girders. For calculation of those spans traditionally used Grillage method when the span structure is analyzed as grid of beams rigidly connected in places of intersections.

In SOFiSTiK such kind of spans might be calculated by approach called T-beam Philosophy which consist of 4 ways of modelling:

1. **Girder in high-rise building**
   Slab modelled by quad elements, T beam inserted as centric beam inside the slab. Center of the beam placed at the center of shell surface. But because in place of intersection we define concrete twice, (it's noticeable in animator by overlapping concrete) there is a need to reduce stiffness of T beam to equivalent I beam cross section, what SOFiSTiK performs automatically by T-beam philosophy shown on Figure 53, that is also working for composite sections if they inserted as centric.

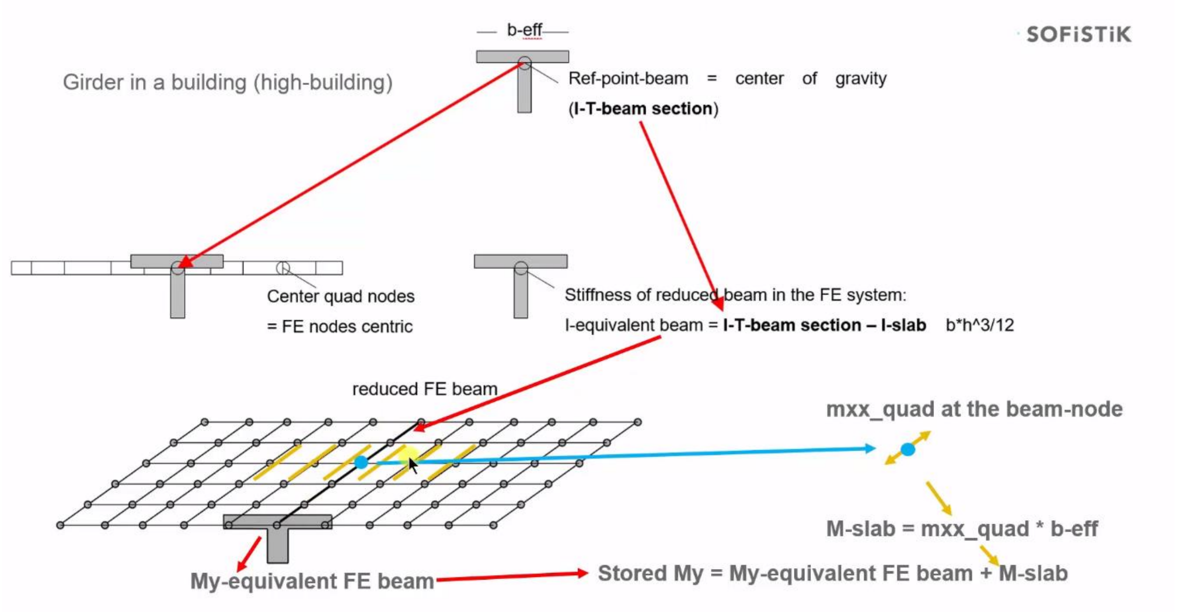


Figure 53. T-Beam philosophy for high-rise buildings (SOFiSTiK AG)

Advantages of this methods is correct total bending moment in beam element for correct sections design. As a resultative area of top layer reinforcement – the biggest value from beam element or from quad element should be taken, depending which one is bigger.

2. **Real system**

   Idea of that method shown on Figure 54 is in discretization as a real system. Slab is modelled by shell elements, and web part modelled as a section of only "extra" part from real slab and defined eccentrically. Or both elements are modelled by quad elements.

   Disadvantage of this method is that we get separate results for slab, and for web part. Thus, we cannot analyze distribution of stresses in the whole section, and cannot make design for shear force for stirrup rebar area.

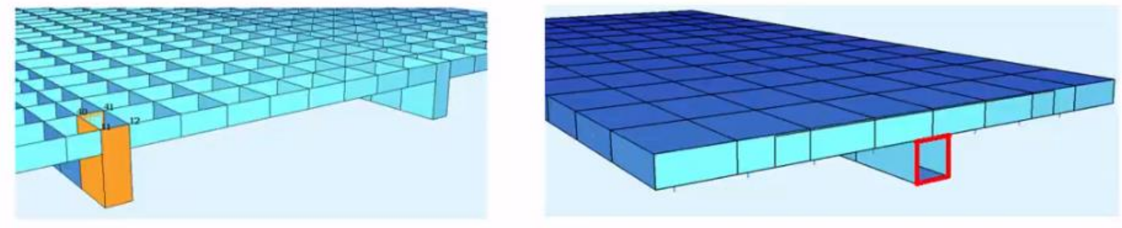


Figure 54. Real system (SOFiSTiK AG)

3. **Orthotropic bridge deck**

   This method shown on Figure 55 is mostly used by bridge engineers. Deck part modelled by shell elements, and main longitudinal superstructure modelled by excentric T beam elements this way that cantilever parts of T-beam cover the whole bridge width.

   To correct weight and stiffness caused by intersections of slab and beam sections, we need to reduce stiffness of slab to 5% from original by Teddy command. As result, we get only T beams working together by transverse load distribution from quad elements, and stiffness of them is ignored for design.
   As an advantage, we get very good Bending moment result, and correct design of beam elements.

   But in case of excentrically loaded beams, we get all the load transferred to beam element, while in real system this load would be taken for bending by slab.

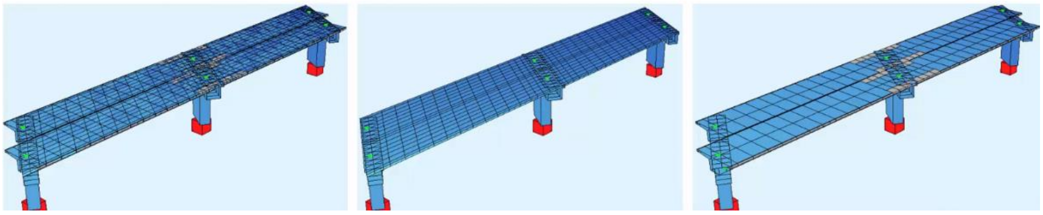


Figure 55. Orthotropic bridge deck (SOFiSTiK AG)

4. **Excentric T beam**

In the new T beam philosophy shown at Figure 56, became available at SOFiSTiK 20. Beam element is modelled by excentric T sections, and slab by quad elements as previously. Corrections of stiffness performed by inserting a “correction beam” by TBEX command in Teddy that compensate double stiffness of quad and beam.

Advantage of this new method is that we have correct stiffness of beam for design, and correct quad model. But it requires extra text command in Teddy.

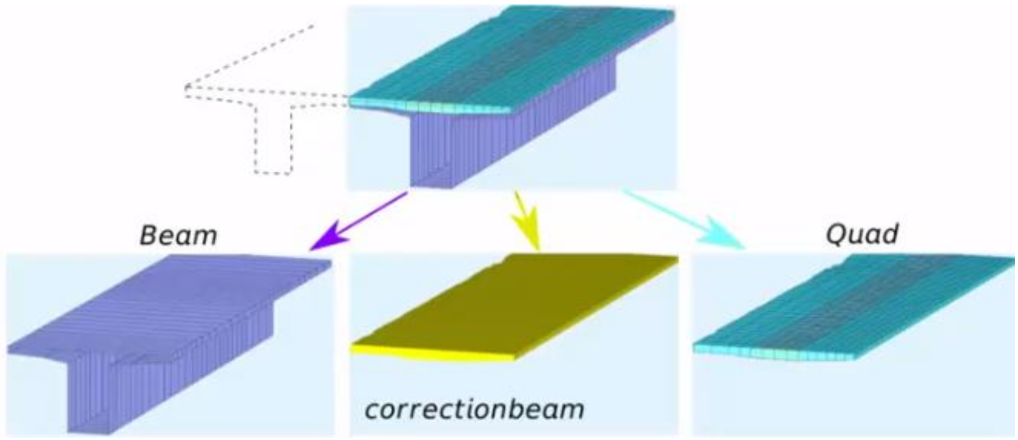


Figure 56. New T-Beam Philosophy

This feature was explained in webinar by Dr. Juergen Bellmann, SOFiSTiK, 2 Feb 2020.
https://www.youtube.com/watch?v=GoT1f0aGdgs&feature=emb_rel_end

5.3.2 Test in SOFiSTiK

To show the example of this feature, there is was TT slab (or bridge deck with 2 main beams) modelled by different methods in Revit:

1. TT slab by T beams cross section + slab by shell element with 2 separate loadceses: evenly distributed area load of 5 kN/m^2 , and equivalent load by line loads applied to beam element only and shown on Figure 57 and 58.

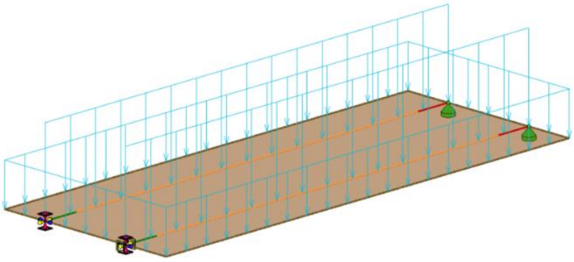


Figure 57. Analytic model

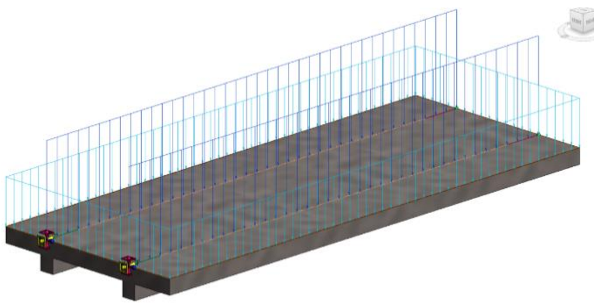


Figure 58. 3D model

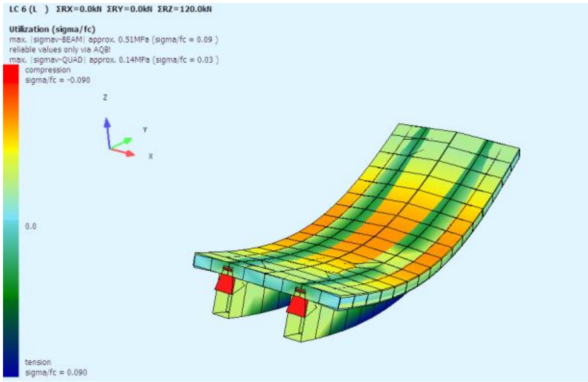


Figure 59. Stress and deflections by area load
max. sigma-beam approx. 0.51MPa
max. sigma-quad approx 0.14MPa

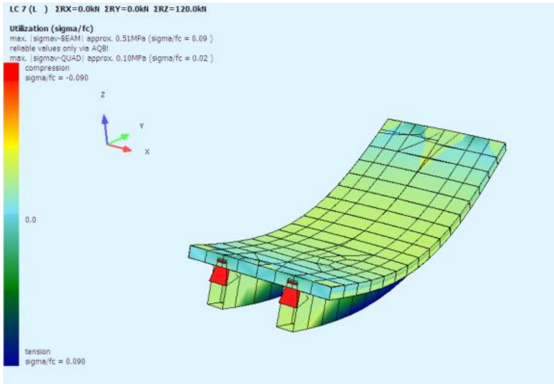


Figure 60. Stress and deflections by equivalent line loads
max. sigma-beam approx. 0.51MPa
max. sigma-quad approx 0.10MPa

The different sigma in quad element, coming from cantilever parts, which are being loaded giving extra moment in x axis direction.

But deflection, stress and moments in the beam element are absolutely the same in two different loadcases.

2. T beam with equivalent line load

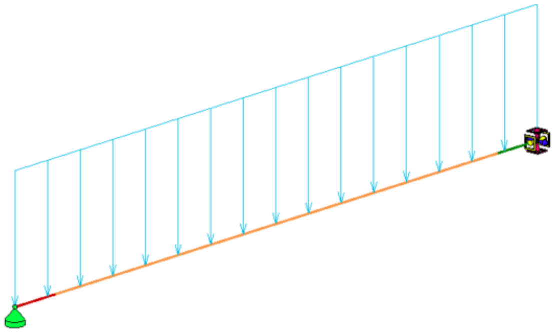


Figure 61. Analytic model

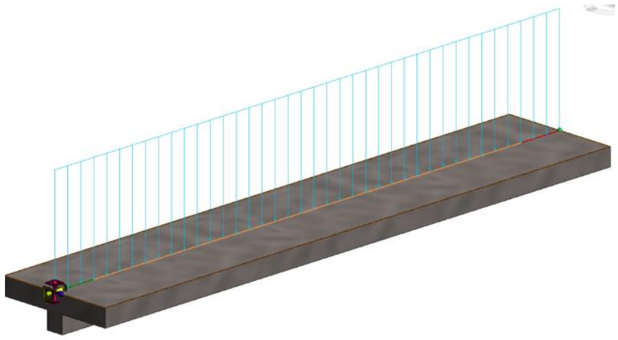


Figure 62. 3D model

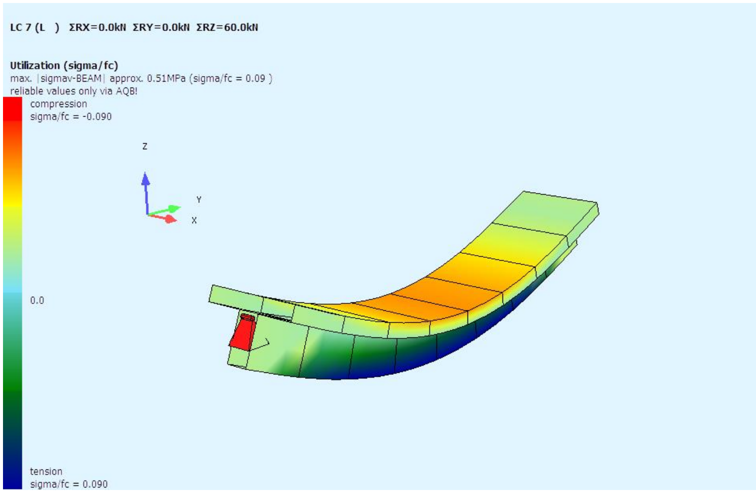


Figure 63. Stress distribution

max. sigma-beam approx. 0.51MPa

We can see on Figure 63, that maximum sigma, and bending moment is the same with the case when we used T-section + Slab for each beam, so the princepe of “T-beam phylosophy” in SOFiSTiK doesn’t take in account rigidity of slab element, but still allow us to recognize it in analitic model like a slab element and apply area load and transfer it to beams. So when we use Revit modelling tools, it seems it’s working as case n.3 - otrhotropic bridge deck method.

3. Web part by centric I beam cross section + slab by shell element

   The same loads were applied to the structure – area load on the slab, and equivalent line load on the beam centreline.

   Without changes to analytical alignment in Revit, T beam cross section from the first testing model was replaced by I beam cross section.

   In this case analytic and 3D physical models looks absolutely the same, but calculation result is the difference. As we can see, in Figure 64, SOFiSTiK doesn't adjust the cross sections below eccentrically so it would look like a correct slab. All cross sections here are centered as it's modelled in analytic model, and gives not correct result.

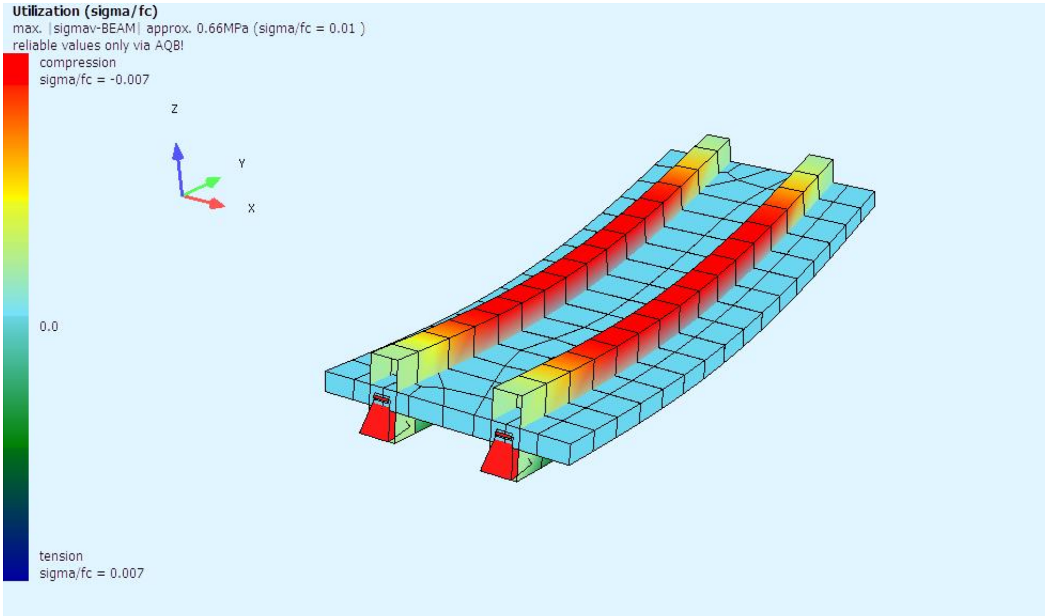


Figure 64. Stress distribution

To solve this issue we should rather use T beam as centric cross section inside the slab, or rather I beam as eccentric , with cross section characteristic of only extra web part. Then this system will work as a case n.2 – real system.

5.4 Pile foundations, and soil simulations

In case of rigidly connected piles to the rock, soil reaction on the grillage part can be ignored and should be calculated as multi-span continuous beam.

Soil simulations must be considered for pile parts, and might be simulated from displacement by spring modulus, linearly applied by pile length or soil conditions should be set up in Bore Profile task of SSD.

In case of Cast in situ foundation with monolithic wall, test of two models was performed to see if wall part does affect on the rigidity of structure and beam design and shown on Figures 65 and 66.

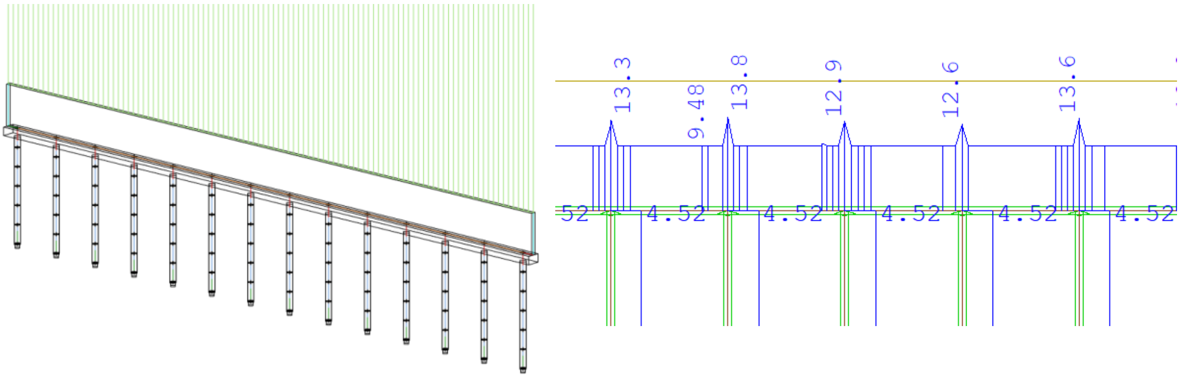


Figure 65. Area of reinforcement, cm^2. Model with wall part.

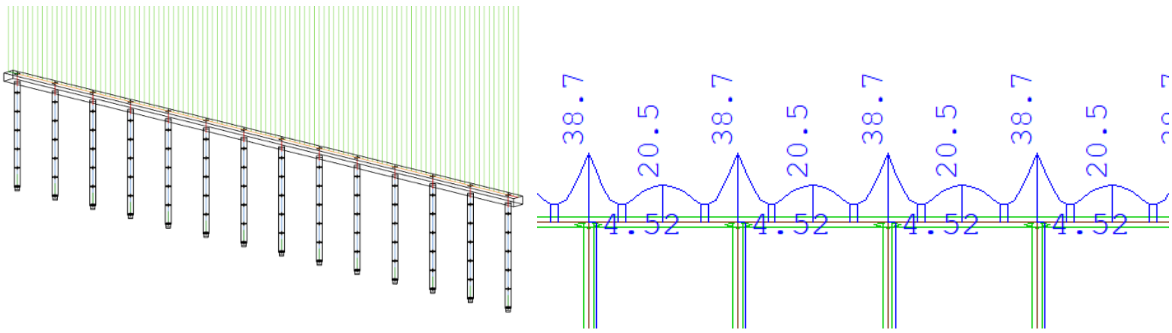


Figure 66. Area of reinforcement, cm^2.Load applied directly to strip foundation and wall part is not modelled.

It can be concluded, that monolithic wall part transfers the compressive zone higher into wall part, because the section in that case looks like flipped T beam. So the bending doesn't affect too much in reality on the design of footing section in that case.

5.5 Peak smoothing

5.5.1 Eurocode reference

According to EN 1992-1-1, Chapter 5.3.2.2 for multi span beams and slabs, there is an option to reduce the peak over the support, if the support has certain thickness.

5.5.2 TEDDY solution

SOFiSTiK , besides SOFiPLUS pre-processor, has TEDDY text editor as alternative native solution. Technically, it's possible to write any kind of task for modules and make changes in already existing ones. For all basic tasks which might be performed by SSD interface, it's possible to write a code. Moreover, for those ones which performed through UI, there is macros that looks like a template of code for certain task.

Since some tasks are possible to perform only through TEDDY text editor, it's important to know how to use it. Full description of all commands are available by pressing F1, and clicking on the name of command. Manual part shown on Figure 67.
Below are some examples of most common codes that will help to start the workflow:

prog _*name of running module*

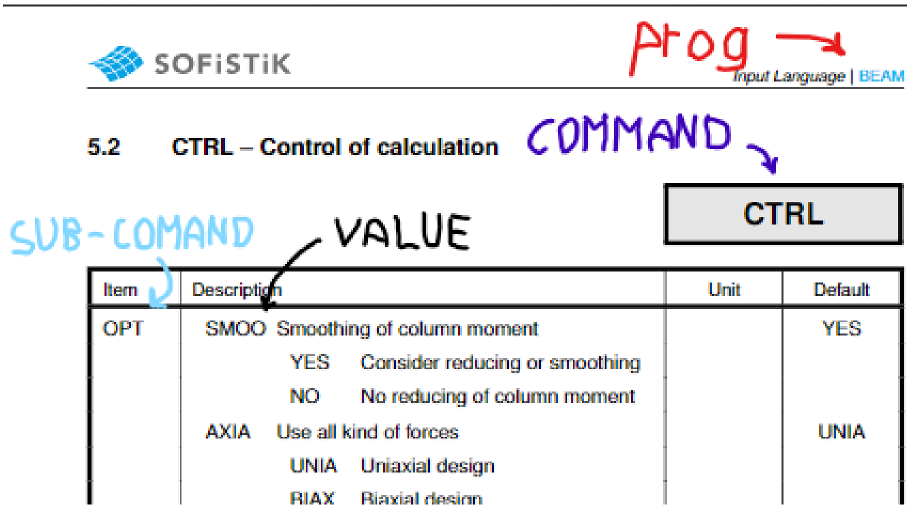
head _*name of the task description*

norm _*name of norm*

!#!Chapter _*chapter text*

$ _*comment text* (might be used for comment to other workers, or for quick deactivating of command by putting $ before code)

end (end of the command, generates automatically)


SOFiSTiK Input Language | BEAM

5.2 CTRL – Control of calculation

CTRL

Item	Description	Unit	Default
OPT	SMOO Smoothing of column moment		YES
	YES Consider reducing or smoothing		
	NO No reducing of column moment		
	AXIA Use all kind of forces		UNIA
	UNIA Uniaxial design		
	BIAX Biaxial design		

Figure 67. Command manual

There is COMMAND (parameter) and SUB-COMMAND (item) goes in the same line. Not every time name of sub-command must be written. In case when you set values for all sub-commands in right turn order, you can just write the value without name of sub-command.

Beam Element created automatically in SSD for beams doesn't recognize support faces for peak smoothing. So, although this task suits for single – span beams, it doesn't work correctly according Eurocode for multiple span continues beams if we want to optimize reinforcement. It recognize columns as point support so we get sharp peaks of the bending moments there. On Figure 68 not smoothed Bending Moment diagram over the support is shown.

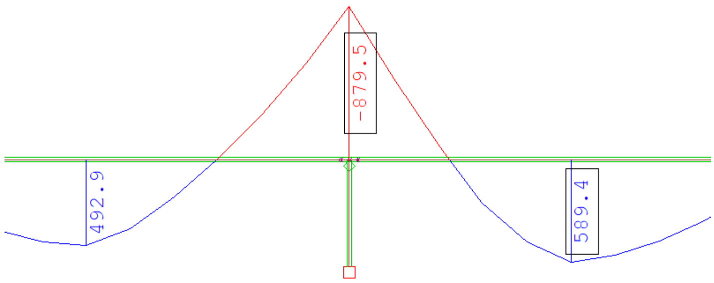


Figure 68. Bending moment over support.

So there is a need to create **Design Element** for this task, but SSD has no graphical interface for module DECREATOR. So, in SSD interface it's possible to write only by TEDDY and write support face locations manually.

1.) First of all we need to create **Design Element** along the structural line using the module DECREATOR to adjust boundary face conditions. And the properties of boundary condition will be written – width and type. So we get a new Design Element. Used code commands are shown on Figure 69 and full workflow in SSD shown on Figure 72.

```
+prog decreator urs:9.2
head  Design element
dsln no 1001 hdiv 0.1 fref sc  $number of structural line
dgeo opt sln id 1001
dsel beam

dslc ref strt s 11.1 typm hfac   $start point face of support and type
dslc ref strt s 11.5 typm hfac   $end point face of support and type

lc 1
end
```

Figure 69. Decreator code in Teddy

2.) After we got Design Element, we need to run calculation **Design ULS (Design Elements).** We can see the difference between not smoothed moment reinforcement at Figure 70 and with smoothed moment reinforcement over support at Figure 71.

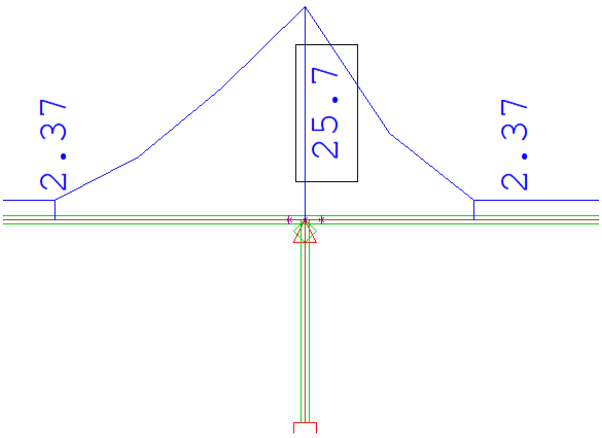


Figure 70. Area of top longitudinal reinforcement [mm^2] after **Design ULS – Beams**. Support conditions doesn't affect on moment reducing. Extra reinforcement appears according to peak value.

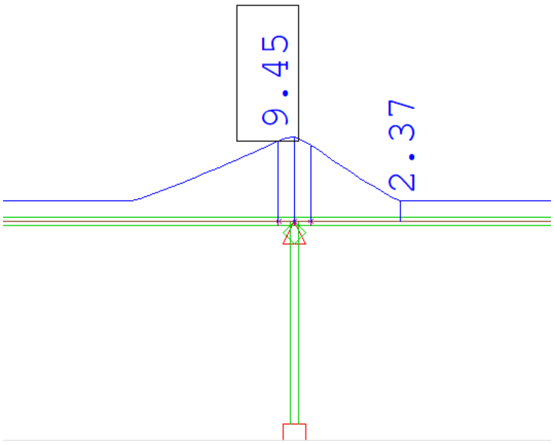


Figure 71. Area of top longitudinal reinforcement [mm^2] after **Design ULS (Design Elements)**. Support conditions affects on the moment reducing and no extra reinforcement appears.

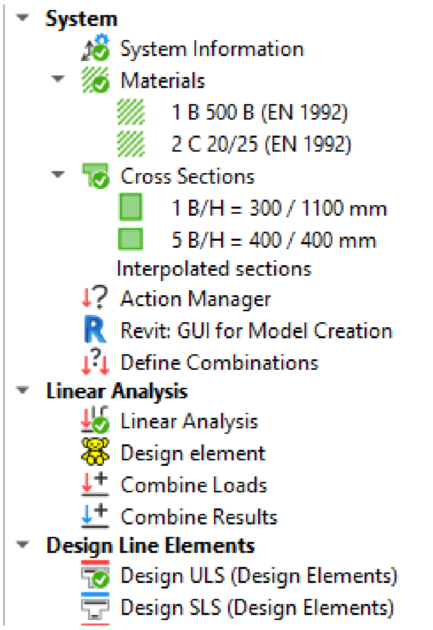


Figure 72. Workflow example in SSD

```
+prog beam urs:9.4
HEAD Peak smoothing
echo full full            $full calculation
CTRL SMOO YES             $Turn peak smoothing on
DSLN NO 1001              $for design element №
COMB 801 (D) "1.35G(1)"   $Choose combinaton from "Combine results"

end
```

Figure 73. Beam code in Teddy

But if we need to provide the **Bending Moment** diagram with smoothed peak, we need to create a command BEAM. But in design, it's enough to get correct reinforcement area. Bending moment line might be needed for checking and reports. Code is shown on Figure 73.

5.5.3 Revit plug-in solution

Another option, with add-ons on Revit, there is a new possibility to see the graphical UI for Beam Design. It recognize support faces from the physical model and creates Design Element correctly. Revit UI of following task is shown on Figure 74.

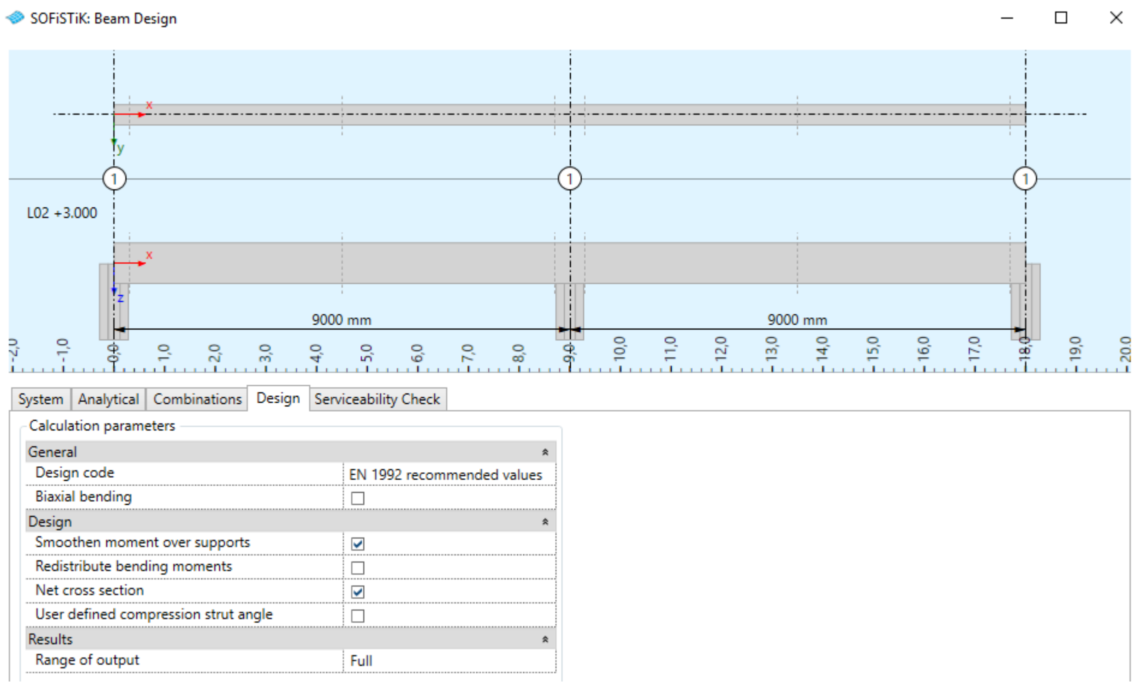
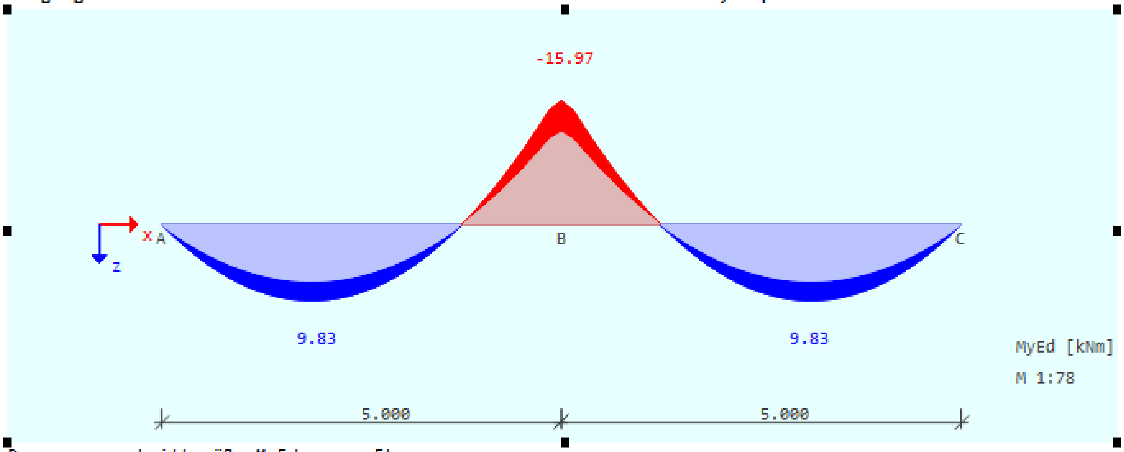


Figure 74. Design of beam element in Revit.

After the calculation, it gives report document that is available for export in Word format, and it's also possible to see the smoothed bending moment diagram for created Design Element. Report is shown on Figure 75.

Biegung mit oder ohne Normalkraft und Normkraft allein EN 1992-1-1, Kap. 6.1



Bemessungsschnittgröße MyEd - x-z Ebene

Stützmomente

Achse	Lager	x [m]	MyEd [kNm]	d [m]	xsi	As,z- [cm2]	Komb
	A	0.000	-0.00	0.460	0.000	2.37	1003
	A,r	0.150	1.51	0.460	0.243	2.37	1003
	B,l	4.850	-14.73	0.460	0.094	2.37	1003
	B	5.000	-15.97¹	0.460	0.094	2.37	1003
	B,r	5.150	-14.73	0.460	0.094	2.37	1003
	C,l	9.850	1.51	0.460	0.243	2.37	1003
	C	10.000	0.00	0.460	0.000	2.37	1003

¹ Ausrundung der Stützmomente nach EN 1992-1-1, 5.3.2.2 (3),(4)

d statische Nutzhöhe

xsi bezogene Druckzonenhöhe xi = x/d

Figure 75. Bending moment diagram. (SOFiSTiK AG)

So, this way Revit as pre-processor gives opportunity to design elements in SOFiSTiK easier and faster.

6 Workflow result

6.1 Overview

The workflow, tested in following research and shown on Figure 76, allows to unite separate tasks in a single BIM working process. Using SOFiSTiK BIM database, and Revit's analytical model it's possible to make a direct, good-working link between FEM and BIM. Reducing the amount of drawing work, and aiming avoidance of double modeling procedure (for analytical model in FEM, and physical model for BIM) it is possible to increase the speed of work and accuracy.

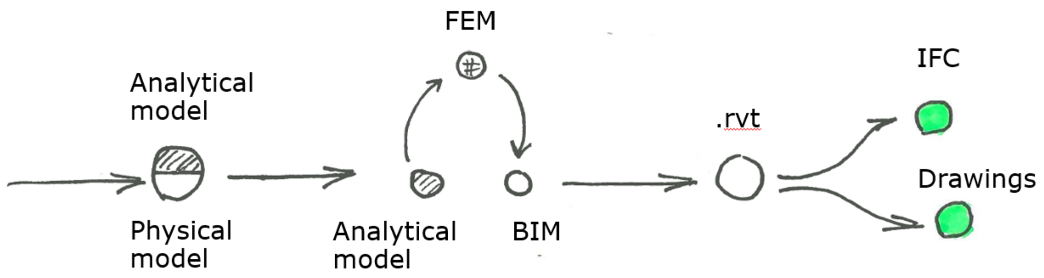


Figure 76. BIM line-based workflow with composite model

It gives the biggest advantage and approximately 1.5 times faster and 2 times better design for structures with relatively simple straight geometry such as columns, walls, slabs, foundations. In practice, that's usually the biggest part of projects.

Even calculations that aren't considered as simple cases such as

- by stage phases,
- with soil simulations,
- nonlinear material behavior
- post tension

can be easily realized in 3D FEA and the final result will be accurate enough for final solution by following steps:

1. Composite model created in Revit by native tools, with constant monitoring and checking analytical model.
2. Load cases are applied, DOFs of nodes and boundary conditions are defined in Revit.
3. Export to SSD, analysis and design.

The following workflow is giving significant improving of design especially on initial stage of project. Because it's easy to collet all the initial data in one platform, easy to make changes and get the detailed FEM model without extra work on it for simple structures. But it requires high control of model creation, and the executive engineer must know FEM basic principles as well as have good skills of modelling.

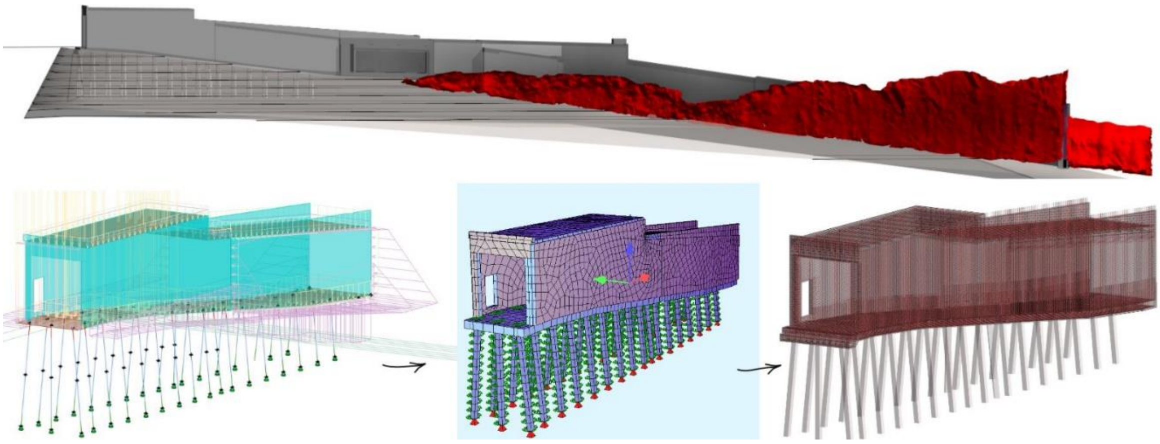


Figure 77. Maintenance tunnel example. Workflow from Analytical model, through FEM model to reinforcement model.

For more complicated geometry, with double directional curve axis lines, or with changing cross-sections and radial curved slabs - improvement of time spent on project is not that much significant so far the visual programming interface Dynamo isn't implemented in modelling process. Because only basic modelling tools are able to create analytical model for export to FEM, there is a need to use the second pre-processor for adjustment of analytical model in difficult parts. In this case, the following workflow looks this way:

1. Physical model is modelled in Revit. Parts of complicated geometry can be handled by mass-form, generic models or adaptive parametric models with help of Dynamo visual programming interface.
2. Model is exported to SOFiPLUS once. We get walls, columns, slabs geometry and placement, that require only few adjustments for high-detailed FEM model.
3. Further detailing modelling of difficult parts and loading should be done in SOFiPLUS
4. Export to SSD, analysis and design.

Among the alternatives, Revit analytical model can be exported to Robot Structural Analysis as well for FEA. And physical model can be exported in IFC or DWG, that is supported by any BIM viewer and CAD program.

This way, export of model to FEM works efficient and without a loose of data. The only limitation in that task – is limitation of modelling possibilities.

6.2 Data transfer back to Revit

Talking about two-directional data flow, there is two options:

1. New feature in 2021 version shown on figure 78 - getting the visualisation of results in Revit model such as FEM mesh, load distribution, stresses, reinforcement area etc. It's possible to make cross cut in slabs as well, and show the results diagrams along the line. Reinforcement is modelled by native Revit tools according to result visualization.
   This is simple, but effective method since you are controlling the whole process of rebar creation, and no IFC rebars created. This feature is included in basic Analysis + Design addon pack.

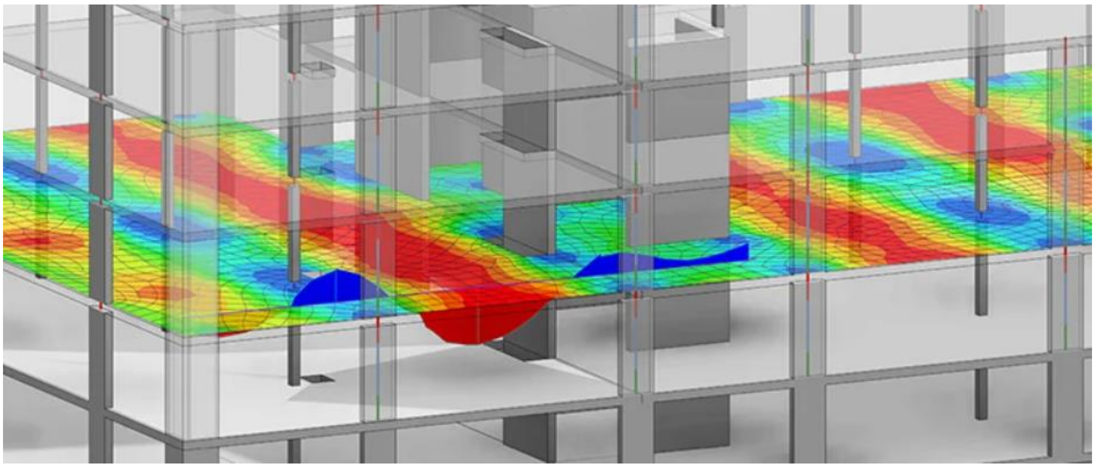


Figure 78. Bending moment distribution and diagram in Revit model. (SOFiSTiK AG)

2. Reinforcement generation plugin shown on Figure 79 is able to read data from .cdb file, and generate reinforcement according to need area from design task for beams, slabs and walls. It allows to create reinforcement for simple structures very fast and easy, but for cast-in situ structures where a lot of rebars and stirrups are connecting elements, there is a need of manual adjustments. Following separate addon cost 1700 USD per year in addition of basic pack.

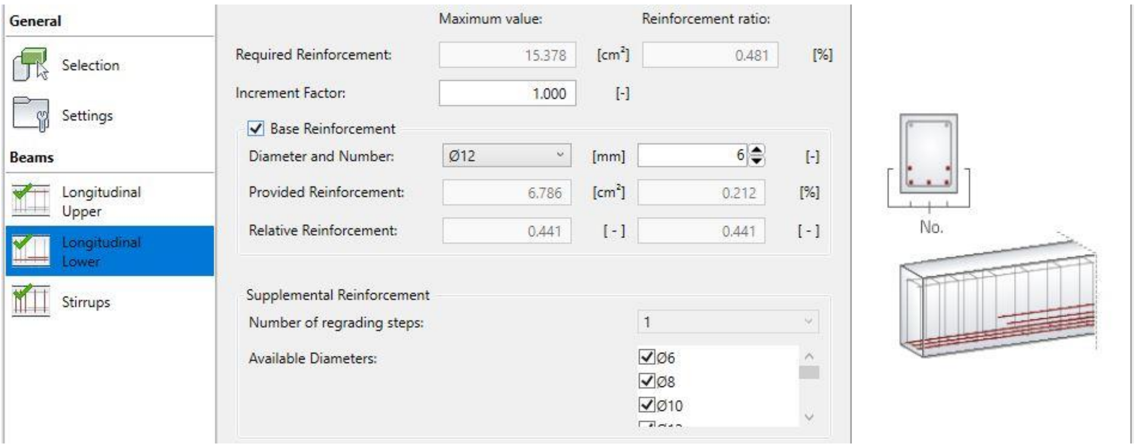


Figure 79. Rebar generation according .cdb data

Also, it’s not necessary to get data back right in Revit. It still possible to adjust the design and model reinforcement in Revit having a look at isolines of reinforcement area in WinGraf file. Because reinforcement design should be easy to do on site, have constant spacing, and also stirrups and rebar ends connections. So in most cases it is enough to take one look for maximum value, compare it with required minimum reinforcement and define needed rebar mesh.

6.3 Drawings

Automatization of drawing creation performs through Reinforcement detailing plug-in and allows to create sheets with drawing representations according to German standards by default. It has a lot of similarities with Finnish standards, but there is also some significant differences.

As outcome from the plugin, we are able to get good bended rebar shape images, marks and detailed bar bending schedule that can be exported also in .abs format directly for bending machines.

But traditional for Scandinavian drawing representation style, arrow ticks on the ends of bars, second layer represented by hidden line must be still done manually or by developing custom families and plug-ins. Detailing examples made by SOFiSTiK Reinforcement Detailing are shown on Figures 80 – 82.

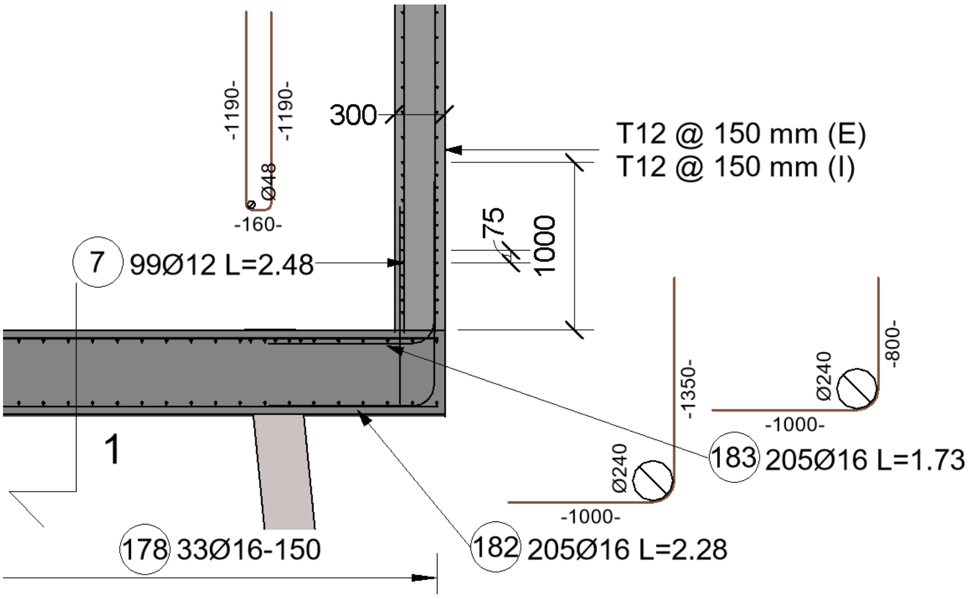


Figure 80. Slab corner reinforcement detailing

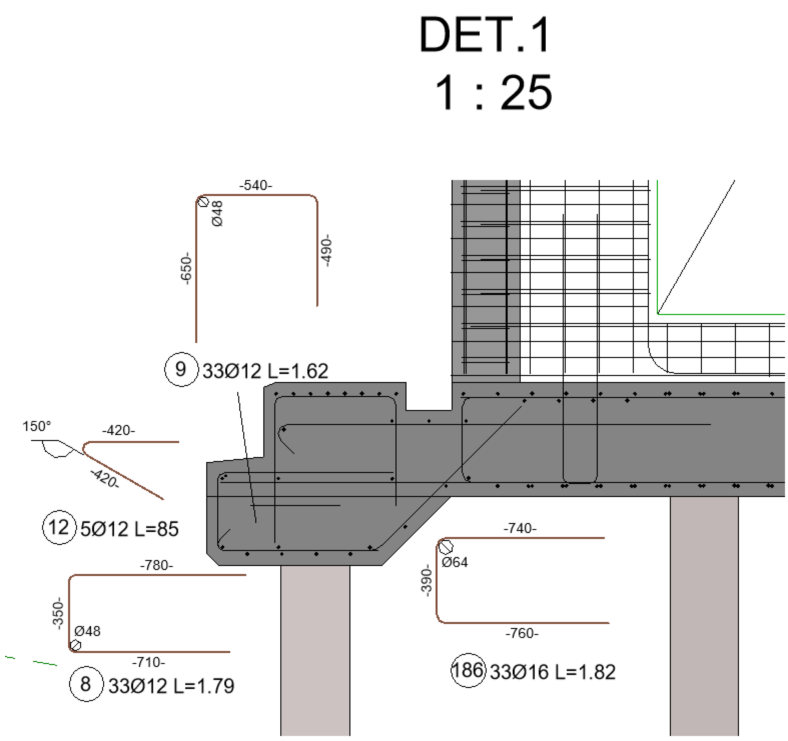


Figure 81. Slab edge reinforcement detailing

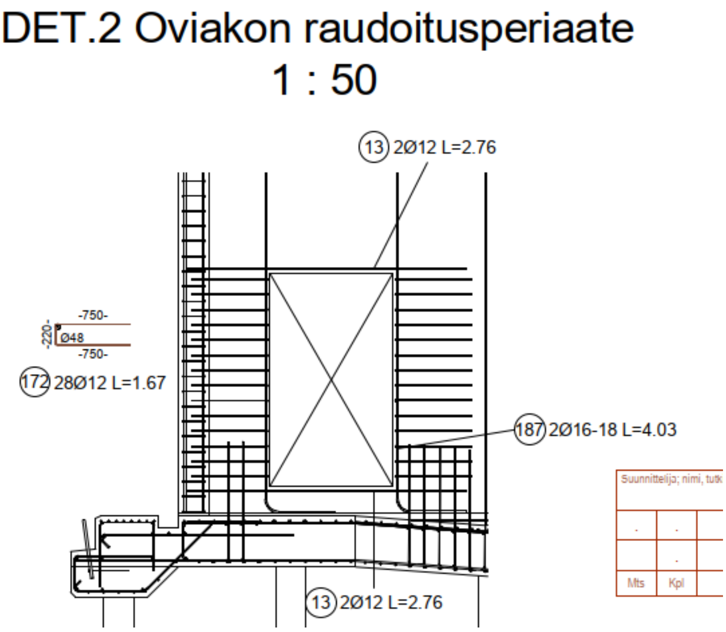


Figure 82. Door opening reinforcement detailing

7 Conclusion

In the following research the investigation of usage Revit+SOFiSTiK workflow for concrete structures was made. Summarize the research results, it is possible to conclude, that following workflow might be used in real cases for future projects.

The limitations that are coming out are not crucial, and it is possible to develop design of project as it is usually required. The following workflow is the most efficient on all stages of projects for cases with simple geometry. Complicated geometry is also possible to handle but with some more assumptions.

Usage of Revit as Pre-Process for SOFiSTiK is usually enough for normal civil engineering tasks. Moreover, export of model to SOFiPLUS is also possible as one-way connection.

As future study, the room for development can be done in learning Revit API, through Dynamo or IronPython code programming for creating custom scripts and plug-ins for automatization of manual specific tasks like applying complicated loads, or polishing the analytical model.

Automatization of poplar tasks can be partly solved by plugins from SOFiSTiK, Naviate, or Graitec developers that obviously significantly improves the traditional workflow. But sometimes in design there is occurring such an important specific things that has no ready solution and problem solver.

References

BIM maturity 2015. The B1M. Accessed 4 Jan 2015
https://www.theb1m.com/video/bim-maturity-easy-as-1-2-3

Lonely BIM, 2019, BIM Dictionary. Accessed 31 Jul 2019
https://bimdictionary.com/en/lonely-bim/1

An Overview of Structural Analysis for Revit, 2018, Revit Structure Blog. Accessed 10 Jan 2018
https://revitstructureblog.wordpress.com/2018/01/10/an-overview-of-structural-analysis-for-revit/

Your Structural model in Autodesk Revit in 10 steps, SOFiSTiK AG

Traffic Loader tutorial. 2020 SOFiSTiK AG
https://www.sofistik.com/documentation/2020/en/tutorials/listoftutorials/general-work-flows/traffic_loader.html

Numerical methods of calculation civil structures. A.V. Lebedev. 2012
Saint-Petersburg State University of Architecture and Civil Engineering
https://www.spbgasu.ru/documents/docs_274.pdf

Basics of Finite Element Method. L.E.Kondratieva. 2007
Vladimir State University
http://www.vlsu.ru/op/fileadmin/Programmy/Aspir-ant/08.06.01/05.23.01/Metod_doc/Method_StroyMech_08.06.01_SM_04.06.2015.pdf

Recommendation for calculating frame of multi-story buildings with rigidity of nodes pre-cast concrete elements. Association "Reinforced Concrete", PhD. Trekin N.N, Moscow 2002
http://www.gostrf.com/normadata/1/4293851/4293851491.htm

Shell shock. Georg Pircher, Jurgen Bellmann. Bd&e 2018 p.59
https://www.sofistik.de/fileadmin/user_upload/Brochures/BDE_shell_shock.pdf

Design of Quad Elements – Layer Design and Baumann Method. Benchmark Example No. 27. SOFiSTiK AG, 2018
https://www.sofistik.de/documentation/2018/en/verification/_static/verification/pdf/dce-en27.pdf

EN 1992-1-1: Eurocode 2: Design of concrete structures

Structural Analysis with the Finite Element Method. Eugenio Oñate. Barcelona 2012

BEMESS manual. Design of Plates and Shells. SOFiSTiK 2020

AQB manual. Design of Cross Sections. SOFiSTiK 2020

BEAM manual. Reinforced Concrete Beam. SOFiSTiK 2020

Bridge Design in SOFiSTiK 2014. Study manual. Yaroshutin D.A 2015

SOFiSTiK. Basics. SOFISTIK AG, 2020

Concrete Plates designed with FEM. Marte Skibeli. Master Thesis 2017, Norway.

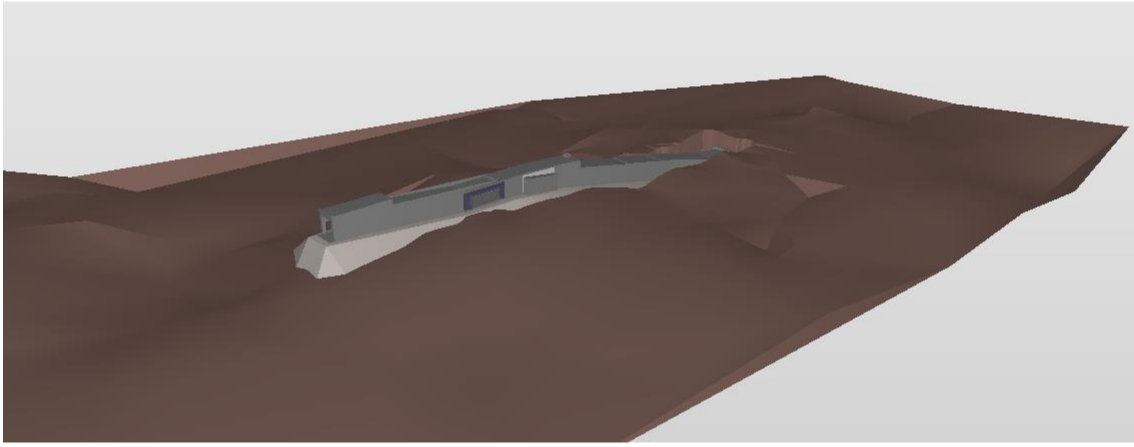
Design of waffle slab with non-linear analysis of concrete. Kleshneva E.I. Master Thesis 2017, Russia.

Getting started with ABAQUS. Database portal.
https://classes.engineering.wustl.edu/2009/spring/mase5513/abaqus/docs/v6.6/books/gsx/default.htm?startat=ch04s01.html

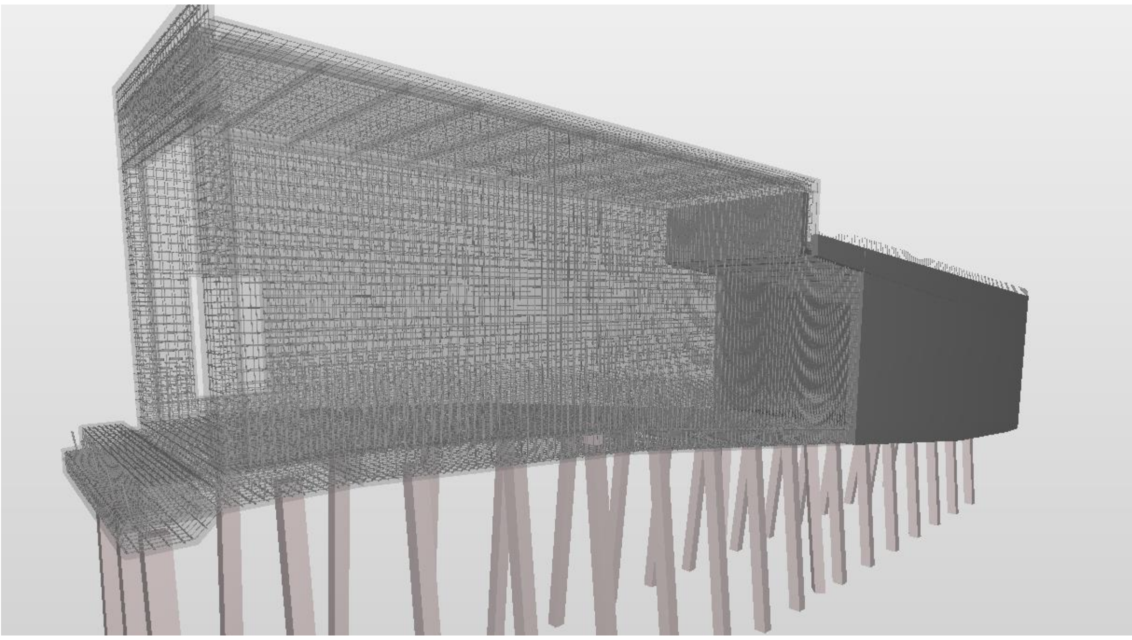
Appendices

IFC models are available by following public link:

https://drive.google.com/drive/folders/1xXeCsDpMwjz1cdFaH26fdmNC7BqYBImJ?usp=sharing

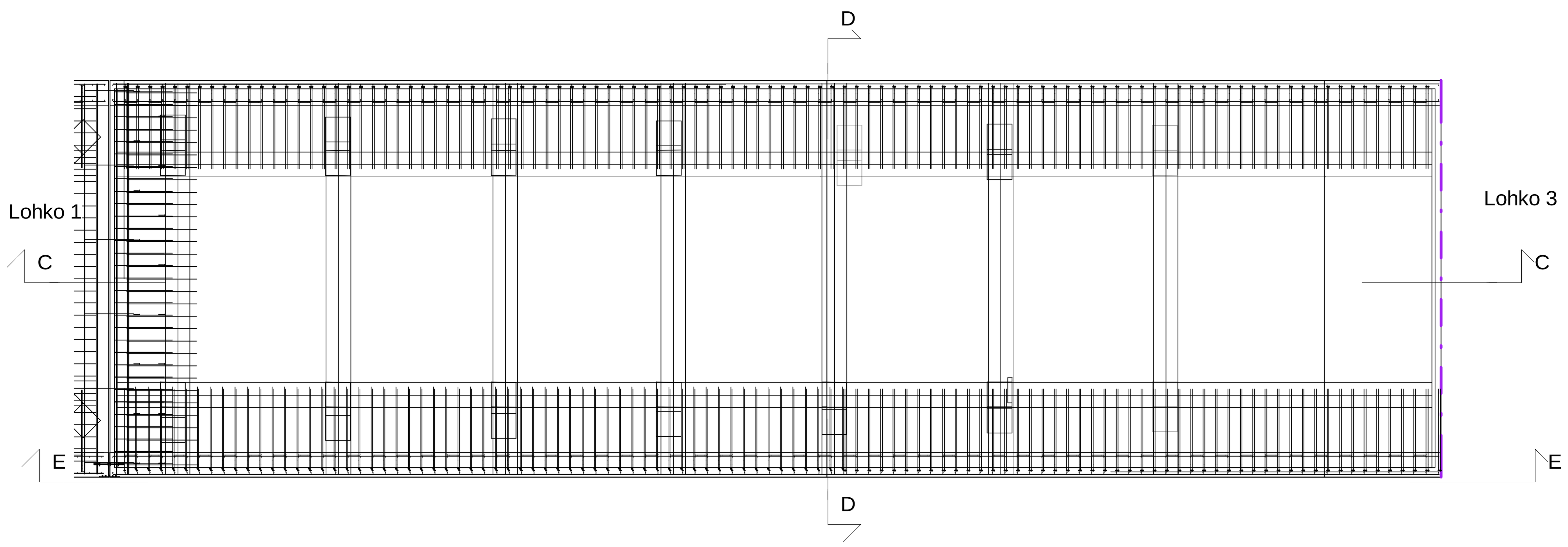


Tunnel_full_geo.ifc – full concrete formwork model with rock surface

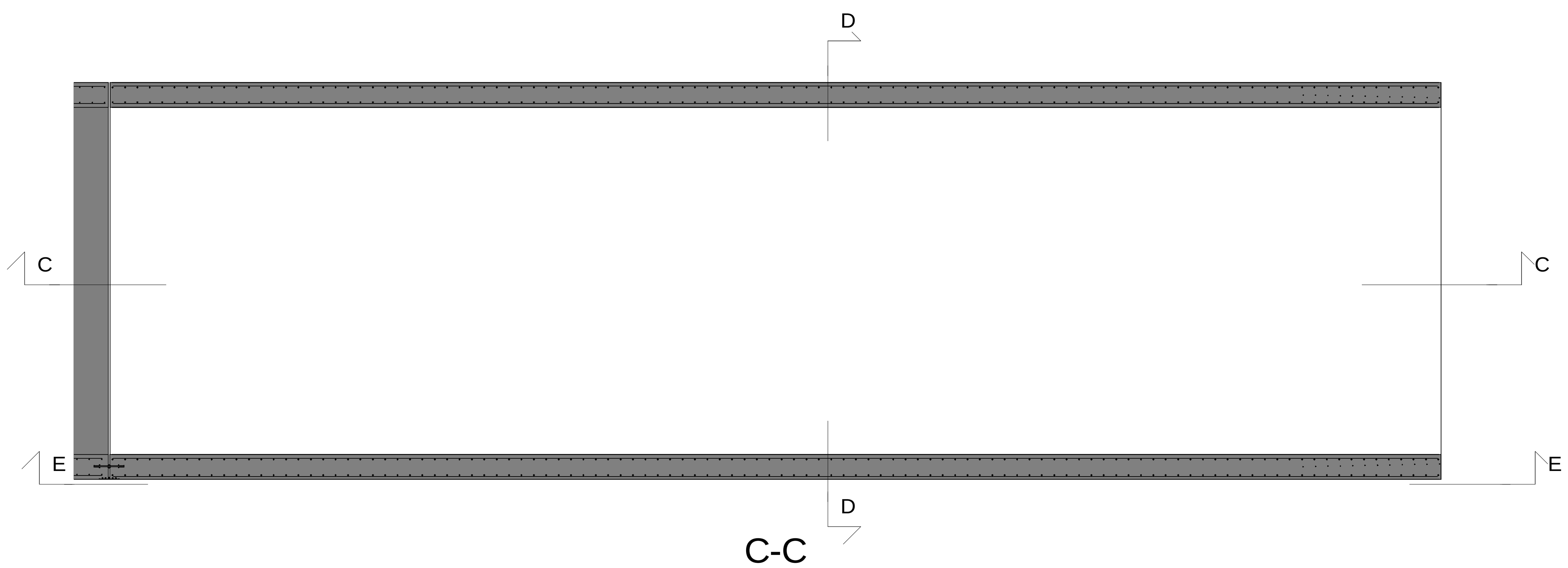


Lohko_1_2.ifc – Isolated reinforcement model of two construction blocks

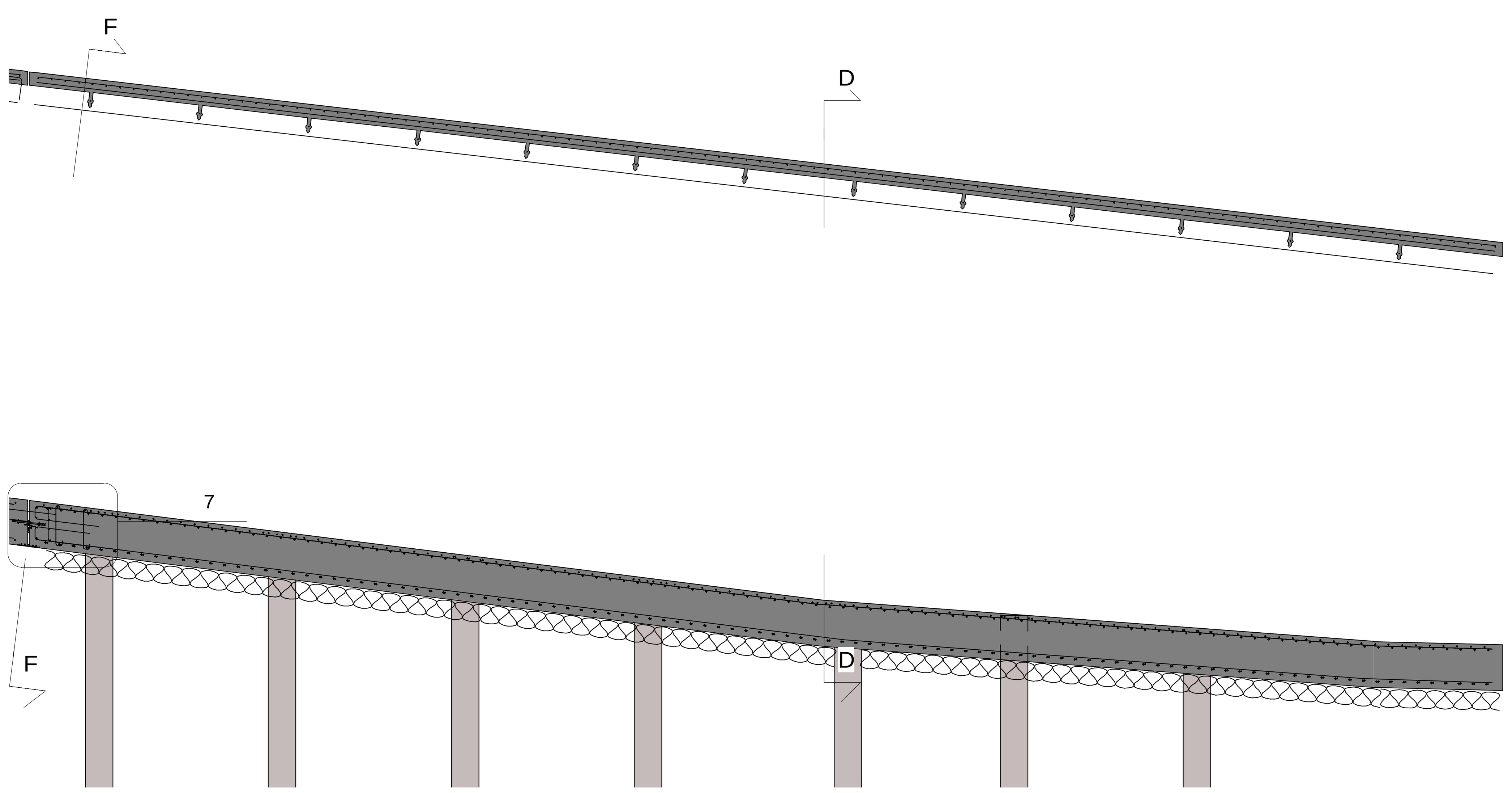
LOHKO 2. POHJALAATAN RAUDOITUS
1 : 50



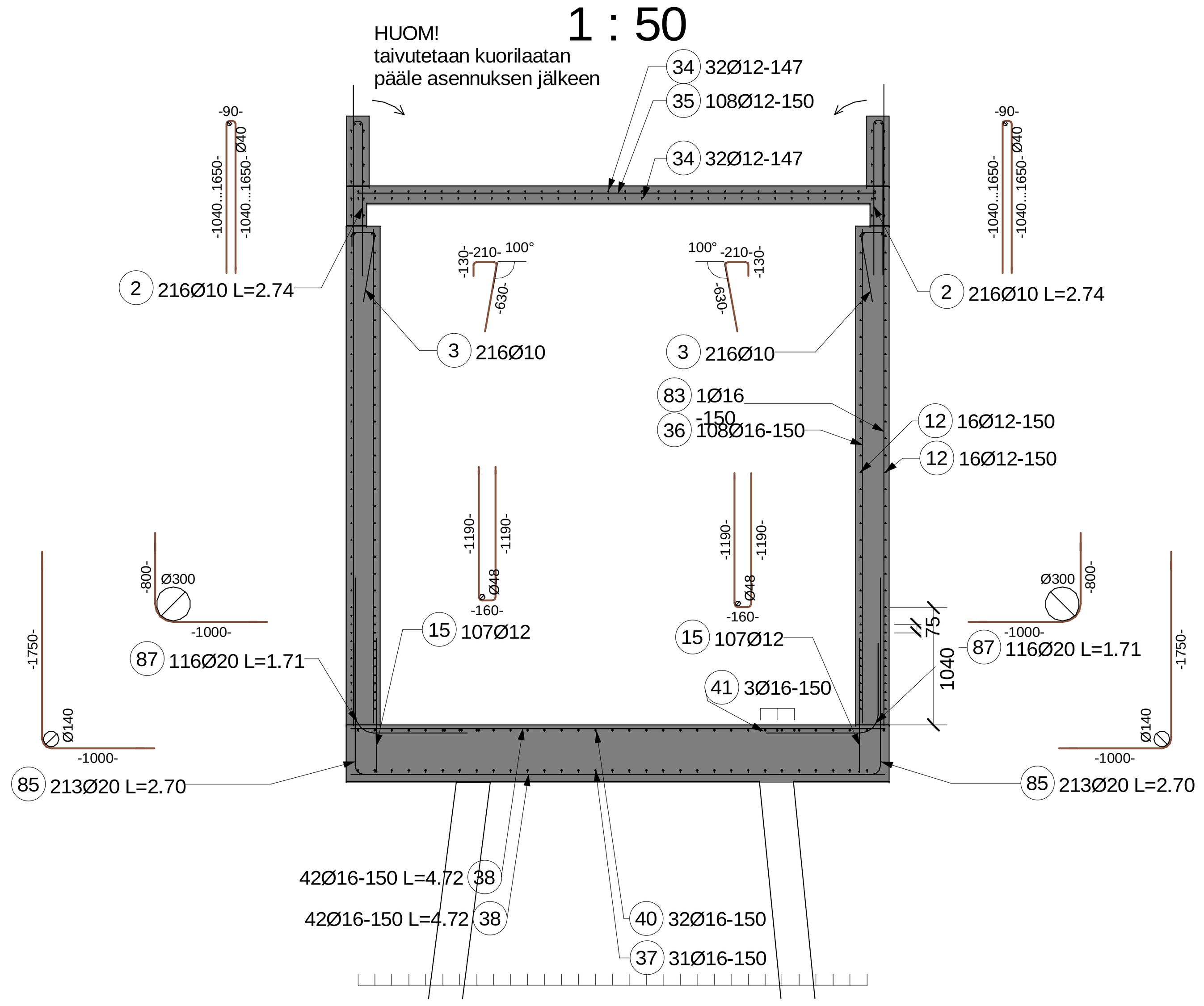
LOHKO 2. HOLVIN RAUDOITUS
1 : 50

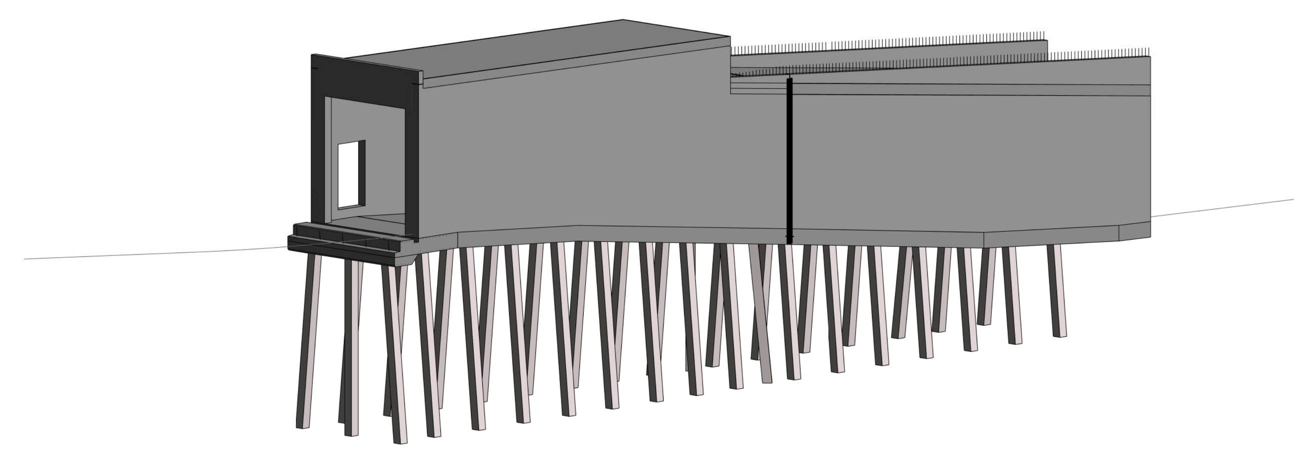


C-C
1 : 50



D
1 : 50

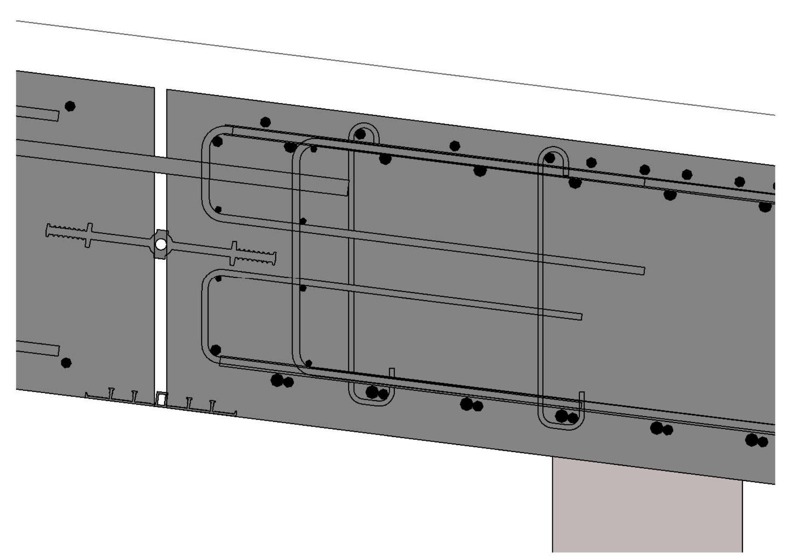




BETONI:
PERUSTUKSET, ULKOSEINÄT JA HOLVI POHJAVEDEN ALAPUOLELLA: C35/45-2
RASITUSLUOKKA: XC3, XF1, XA2
ULKOSEINÄT JA HOLVI POHJAVEDEN YLÄPUOLELLA: C35/45-2
RASITUSLUOKKA: XC3, XF3, XA2, XD1
ULKOSEINÄT JA HOLVI ULKOILMASSA: C35/45-2-P30
RASITUSLUOKKA: XC4, XF3, XD1
POHJALAATTA: C35/45-2
RASITUSLUOKKA: XC3, XF4, XA2, XD1
TASAUSVALU: C25/30
TOTEUTUSLUOKKA: 2
TOLERANSSILUOKKA: 1

BETONIPEITTEEN NIMELLISARVO 40 mm , SALLITTU POIKKEAMA 10 mm

DET.3
1 : 10



TERÄSTEN JATKOSPITUUDET:

T10 lj>600 mm
T12 lj>750 mm
T16 lj>1000 mm
T20 lj>1300 mm
T25 lj>1600 mm

HUOM! VAIN JOKA TOISEN TERÄKSEN SAA JATKAA SAMASSA POIKKILEIKKAUKSESSA

Suunnittelija; nimi, tutkinto ja allekirjoitus	Tark./hyv.	PS tark./hyv.

Mts	Kpl	Pvm	Muutos	Suunn./Piirt.	Tark.
-	-	-	-		
	-				

Kaup.osa/Kylä · Kortteli/Tila · Tontti/Rn:o	Viranomaisten merkintöjä
Rakennustoimenpide	Piirustuslaji · Juoks. n:o
Rakennuksen numero/Rakennusten numerot/Rakennustunnus/Rakennustunnukset	Korkeus-/Koordinaattijärjestelmä · Rataosa/paaluväli
Rakennushanke	Piirustuksen sisältö · Mittakaava
Suunnittelutoimiston nimi ja yhteystiedot	Suunnitteluala, tunnus ja n:o ja lohko · Muutos
Suunnittelija; nimi, tutkinto ja allekirjoitus	Tiedosto · Projekti n:o 101011361
Piirt. · Tark./hyv.	
PS tark./hyv.	Päiväys

X Y & Partners

Project: Maintenance tunnel | **Schedule ref: LOHKO 2.** | **Date:** 2020-12-07
Client: Owner | **Prepared by: Artem Shatalin** | **Checked by: Checker**

Project

Name: Project Name
Address: Enter address here
Client: Owner
Shape Codes According to: EN ISO 3766:2003
Revision Number:
Revision Date:

The bar length is calculated on the basis of the center line.

Weight Schedule - straight bars

Material: Default				
Sizes used	**Type**	**Mass per Unit Length**	**Total Length**	**Total Weight**
10 mm	Ø10, Dmin=4φ	0.62 kg/m	459.06	284.62 kg
12 mm	Ø 12	0.89 kg/m	3788.30	3371.59 kg
16 mm	Ø16, Dmin=4φ	1.58 kg/m	3230.91	5104.84 kg
25 mm	Ø25, Dmin=15φ	3.85 kg/m	3.60	13.86 kg
TOTALS				8774.90 kg

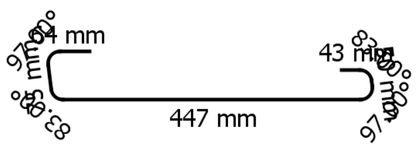
Weight Schedule – bent bars

Material: Default			
Sizes used	**Mass per Unit Length**	**Total Length**	**Total Weight**
8 mm	0.39 kg/m	9.76	3.81 kg
10 mm	0.62 kg/m	833.64	516.86 kg
12 mm	0.89 kg/m	375.41	334.11 kg
16 mm	1.58 kg/m	1102.30	1741.63 kg
20 mm	2.47 kg/m	198.36	489.95 kg
20 mm	2.47 kg/m	744.85	1839.78 kg
TOTALS			4926.14 kg

Rebar Bending Schedule

Member	Bar Mark	Bar Diameter	Length of each bar	Total No.	Total Length	Bend Diameter	Shape Code	Hook Start/End	A	B	C	D	E / R	H_{Start} / H_{End}	Rev
	1	8 mm	0.61	16	9.76	32 mm	99								

Segment	Length	Bending Angle
-A-	43 mm	83.00°
-B-	50 mm	97.00°
-C-	447 mm	83.00°
-D-	75 mm	97.00°
-E-	64 mm	0.00 °



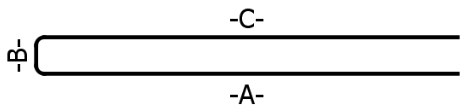
X Y & Partners

Project: Maintenance tunnel
Client: Owner

Schedule ref: LOHKO 2.
Prepared by: Artem Shatalin

Date: 2020-12-07
Checked by: Checker

Member	Bar Mark	Bar Diameter	Length of each bar	Total No.	Total Length	Bend Diameter	Shape Code	Hook Start/End		A	B	C	D	E / R	H_{Start} / H_{End}		Rev
	2	10 mm	2740-AVG.	216	591.84	40 mm	21			VAR.	90	VAR.					



Mark	Quantity	Length	A	C
2.1	2	2130	1040	1040
2.2	2	2140	1050	1050
2.3	2	2150	1050	1050
2.4	2	2160	1060	1060
2.5	2	2170	1060	1060
2.6	2	2190	1070	1070
2.7	2	2200	1070	1070
2.8	2	2210	1080	1080
2.9	2	2220	1090	1090
2.10	2	2230	1090	1090
2.11	2	2240	1100	1100
2.12	2	2250	1100	1100
2.13	2	2270	1110	1110
2.14	2	2280	1110	1110
2.15	2	2290	1120	1120
2.16	2	2300	1130	1130
2.17	2	2310	1130	1130
2.18	2	2320	1140	1140
2.19	2	2330	1140	1140
2.20	2	2350	1150	1150
2.21	2	2360	1150	1150
2.22	2	2370	1160	1160
2.23	2	2380	1170	1170
2.24	2	2390	1170	1170
2.25	2	2400	1180	1180
2.26	2	2410	1180	1180
2.27	2	2430	1190	1190
2.28	2	2440	1190	1190
2.29	2	2450	1200	1200
2.30	2	2460	1210	1210
2.31	2	2470	1210	1210
2.32	2	2480	1220	1220
2.33	2	2490	1220	1220
2.34	2	2510	1230	1230
2.35	2	2520	1230	1230
2.36	2	2530	1240	1240
2.37	2	2540	1250	1250
2.38	2	2550	1250	1250
2.39	2	2560	1260	1260
2.40	2	2570	1260	1260
2.41	2	2590	1270	1270
2.42	2	2600	1270	1270
2.43	2	2610	1280	1280
2.44	2	2620	1290	1290
2.45	2	2630	1290	1290
2.46	2	2640	1300	1300
2.47	2	2650	1300	1300

X Y & Partners

Project: Maintenance tunnel **Schedule ref: LOHKO 2.** **Date:** 2020-12-07
Client: Owner **Prepared by: Artem Shatalin** **Checked by: Checker**

Member	Bar Mark	Bar Diameter	Length of each bar	Total No.	Total Length	Bend Diameter	Shape Code	Hook Start/End	A	B	C	D	E / R	H_{Start} / H_{End}	Rev

2.48	2	2670	1310	1310
2.49	2	2680	1310	1310
2.50	2	2690	1320	1320
2.51	2	2700	1330	1330
2.52	2	2710	1330	1330
2.53	2	2720	1340	1340
2.54	2	2730	1340	1340
2.55	2	2750	1350	1350
2.56	2	2760	1350	1350
2.57	2	2770	1360	1360
2.58	2	2780	1370	1370
2.59	2	2790	1370	1370
2.60	2	2800	1380	1380
2.61	2	2810	1380	1380
2.62	2	2830	1390	1390
2.63	2	2840	1390	1390
2.64	2	2850	1400	1400
2.65	2	2860	1410	1410
2.66	2	2870	1410	1410
2.67	2	2880	1420	1420
2.68	2	2890	1420	1420
2.69	2	2910	1430	1430
2.70	2	2920	1430	1430
2.71	2	2930	1440	1440
2.72	2	2940	1450	1450
2.73	2	2950	1450	1450
2.74	2	2960	1460	1460
2.75	2	2970	1460	1460
2.76	2	2990	1470	1470
2.77	2	3000	1470	1470
2.78	2	3010	1480	1480
2.79	2	3020	1490	1490
2.80	2	3030	1490	1490
2.81	2	3040	1500	1500
2.82	2	3050	1500	1500
2.83	2	3070	1510	1510
2.84	2	3080	1510	1510
2.85	2	3090	1520	1520
2.86	2	3100	1530	1530
2.87	2	3110	1530	1530
2.88	2	3120	1540	1540
2.89	2	3130	1540	1540
2.90	2	3150	1550	1550
2.91	2	3160	1550	1550
2.92	2	3170	1560	1560
2.93	2	3180	1570	1570
2.94	2	3190	1570	1570
2.95	2	3200	1580	1580
2.96	2	3210	1580	1580
2.97	2	3230	1590	1590
2.98	2	3240	1590	1590
2.99	2	3250	1600	1600
2.100	2	3260	1610	1610
2.101	2	3270	1610	1610
2.102	2	3280	1620	1620
2.103	2	3290	1620	1620
2.104	2	3310	1630	1630
2.105	2	3320	1630	1630

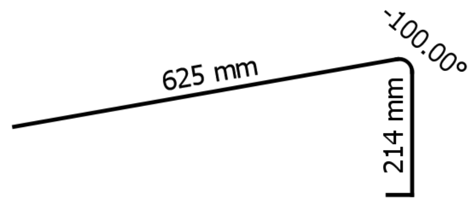
X Y & Partners

Project: Maintenance tunnel | **Schedule ref:** LOHKO 2. | **Date:** 2020-12-07
Client: Owner | **Prepared by:** Artem Shatalin | **Checked by:** Checker

2.106	2	3330	1640	1640
2.107	2	3340	1650	1650
2.108	2	3350	1650	1650

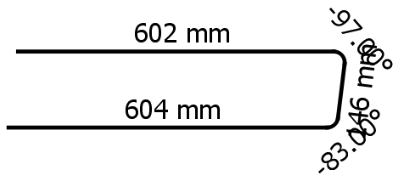
Member	Bar Mark	Bar Diameter	Length of each bar	Total No.	Total Length	Bend Diameter	Shape Code	Hook Start/End	A	B	C	D	E / R	H_{Start} / H_{End}	Rev
	3	10 mm	0.93	216	200.88	40 mm	99	1						130	

Segment	Length	Bending Angle
-A-	130	-90.00 °
-B-	214 mm	-100.00°
-C-	625 mm	0.00 °

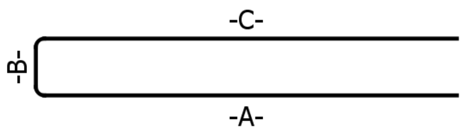


Member	Bar Mark	Bar Diameter	Length of each bar	Total No.	Total Length	Bend Diameter	Shape Code	Hook Start/End	A	B	C	D	E / R	H_{Start} / H_{End}	Rev
	10	10 mm	1.32	31	40.92	40 mm	99								

Segment	Length	Bending Angle
-A-	604 mm	-83.00°
-B-	146 mm	-97.00°
-C-	602 mm	0.00 °

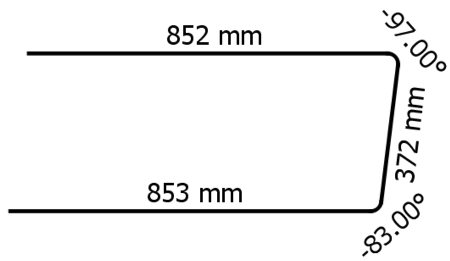


Member	Bar Mark	Bar Diameter	Length of each bar	Total No.	Total Length	Bend Diameter	Shape Code	Hook Start/End	A	B	C	D	E / R	H_{Start} / H_{End}	Rev
	15	12 mm	2.48	107	265.36	48 mm	21		1190	160	1190				



Member	Bar Mark	Bar Diameter	Length of each bar	Total No.	Total Length	Bend Diameter	Shape Code	Hook Start/End	A	B	C	D	E / R	H_{Start} / H_{End}	Rev
	29	12 mm	2.04	31	63.24	48 mm	99								

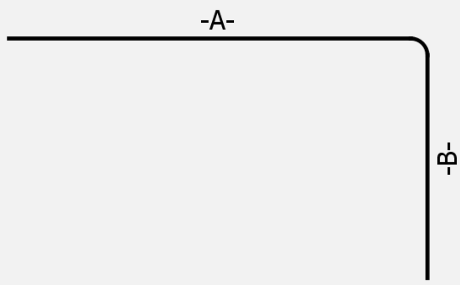
Segment	Length	Bending Angle
-A-	853 mm	-83.00°
-B-	372 mm	-97.00°
-C-	852 mm	0.00 °



Member	Bar Mark	Bar Diameter	Length of each bar	Total No.	Total Length	Bend Diameter	Shape Code	Hook Start/End	A	B	C	D	E / R	H_{Start} / H_{End}	Rev
	30	12 mm	1.51	31	46.81	48 mm	99								
	37	16 mm	15.98	31	495.38	64 mm	25		8880	5690	1420	440	60		

X Y & Partners

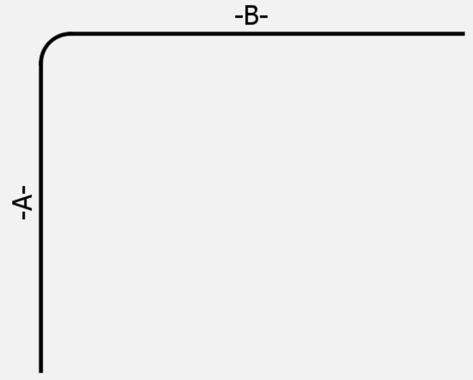
Project: Maintenance tunnel
Client: Owner
Schedule ref: LOHKO 2.
Prepared by: Artem Shatalin
Date: 2020-12-07
Checked by: Checker

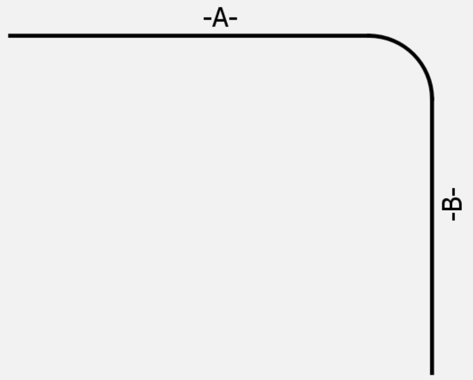
Member	Bar Mark	Bar Diameter	Length of each bar	Total No.	Total Length	Bend Diameter	Shape Code	Hook Start/End	A	B	C	D	E / R	H_{Start} / H_{End}	Rev
															
	40	16 mm	15.97	32	511.04	64 mm	25		8560	6120	1280	430	60		
															
	41	16 mm	15.98	3	47.94	64 mm	25		8570	6110	1300	420	60		
															
	84	16 mm	15.98	3	47.94	64 mm	25		8570	6130	1280	430	60		
															
	85	20 mm	2.70	213	575.10	140 mm	11		1750	1000					
															
	86	20 mm	1.75	97	169.75	140 mm	11		800	1000					

X Y & Partners

Project: Maintenance tunnel
Client: Owner
Schedule ref: LOHKO 2.
Prepared by: Artem Shatalin
Date: 2020-12-07
Checked by: Checker

Member	Bar Mark	Bar Diameter	Length of each bar	Total No.	Total Length	Bend Diameter	Shape Code	Hook Start/End	A	B	C	D	E / R	H_{Start} / H_{End}	Rev
	87	20 mm	1.71	116	198.36	300 mm	11		1000	800					





Author
Artem Shatalin
Phone
Click here to enter business phone.
Mobile
Click here to enter mobile phone.
E-mail
Artem.shatalin@afry.com
Date
01/12/2020
Project ID
Click here to enter Project ID.

Recipient
AFRY

Guideline

Reinforcement design using Revit + SOFiSTiK