BLD125052

From Revit to FEM and back – with use of Dynamo

Anders Hejnfelt
Sweco Denmark

Learning Objectives

- Learn about the analytical model in Revit
- Learn how to correct and control the analytical model with Dynamo
- Apply loads automatically within Revit
- Learn about the advantages of using the analytical model

Description

The class will show a design process where we use a building model in Revit software for structural analysis. We will optimize the design process using Dynamo programming. We will then use Dynamo to correct the analytical model in Revit and apply loads to the structure. After we've exported the analytical model to a finite element method (FEM) program and analyzed it, we will use Dynamo to write calculation results back to the Revit elements where they can be illustrated.

Speaker(s)

Anders Hejnfelt *is working as a structural engineer for Sweco in Denmark. Anders finished his M.Sc. degree in civil engineering in 2017. The main focus of his master thesis was structural design and analysis through visual programming. In his job as structural engineer he is working with implementation of Dynamo tools and development of the use of Dynamo in combination with structural analysis.*

Anders.Hejnfelt@Sweco.dk

Introduction

In Denmark, many companies have focus on BIM implementation and the possibilities when using BIM. Thereby BIM models in Revit are widely used on both small and large scale projects. In Sweco Denmark we are also using Finite Element Models in the structural design phase. Typically the BIM model in Revit and the FEM model are modelled separately leading to a large amount of work when project changes happens and should be implemented in both models.

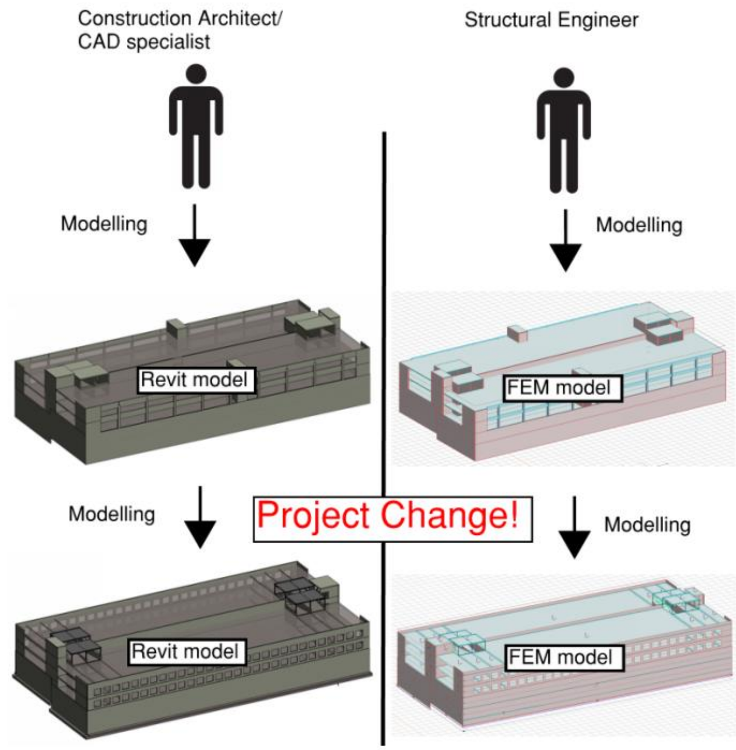


Figure 1: Typical design process today

It is fast and easy to export the model to FEM software as Robot Structural Analysis, but since the analytical elements typically not are connected correctly the model can not be used for analysis. The large amount of manual work to correct the model often seems difficult and it is easier to build a new FEM model from scratch. An example of an exported building model from Revit is illustrated below. It can easily be seen that the analytical model is not connected correctly since there are multiple gaps between the wall elements.

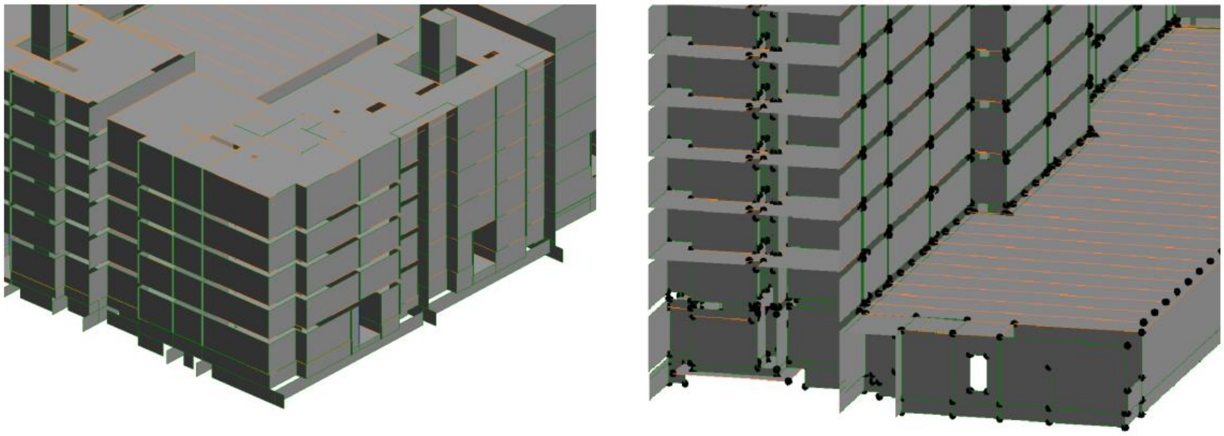


Figure 2: Inconsistent analytical model in Revit

The thickness difference between the structural and analytical model often results in situations illustrated below. It should be noted that this problem often occurs on projects in Denmark because of the wide use of prefabricated elements.

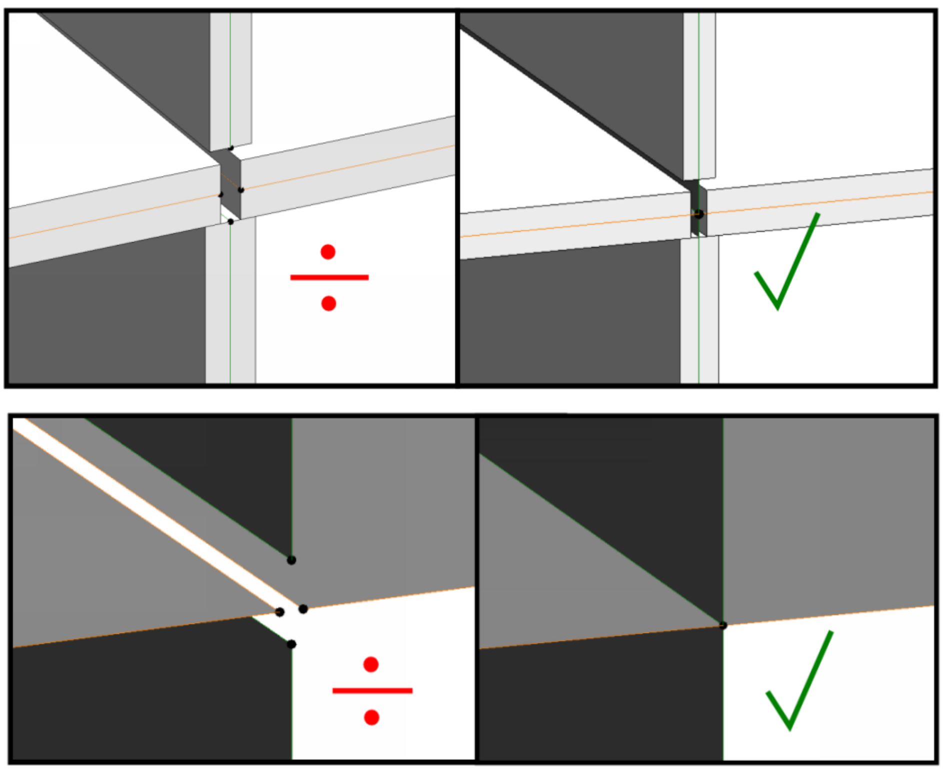


Figure 3: Connection between analytical elements

When modelling prefabricated wall- and floor elements the in-situ cast connections are often not included in the Revit model. Thereby the chance of having a consistent analytical model in Revit will be small.

This class will propose a method to control the analytical model by aligning to gridlines and levels and take advantage of the possibilities with Dynamo. Thereby a typical structural design workflow can be changed to the illustration below.

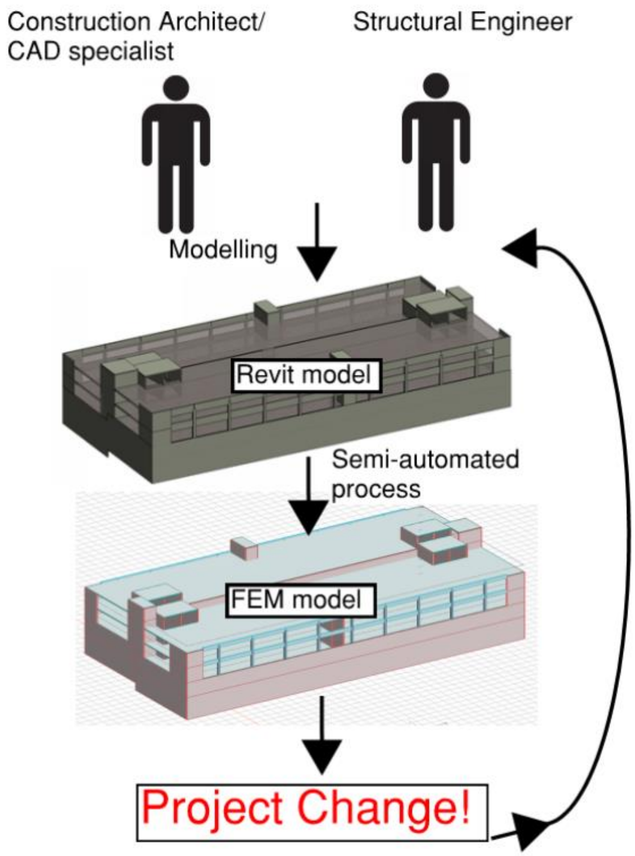


Figure 4: Optimized design process

When project changes are introduced the geometry from Revit can be updated and modelling time in the FEM program have been saved. Furthermore, it ensures consistency between the two models. It should be noted that the update function is very dependent on the interoperability between Revit and the FEM program, since settings such as line connections should be kept in the FEM program.

Correction of analytical models

Alignment method

In connection to the structural model in Revit is an analytical model consisting of the elements illustrated in the figure below.

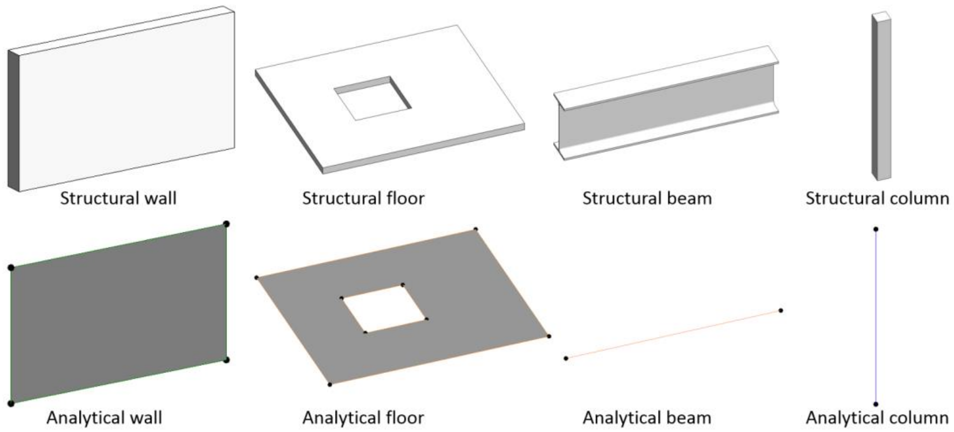


Figure 5: Analytical elements in Revit

In the properties for the analytical elements are the alignment settings where it is possible to control the analytical model. As an example, it is possible to align a wall to a grid line horizontally and a level in the top and bottom of the wall. Thereby the wall will be locked into the grid system. Alignment to reference planes and levels are also an option.

The alignment properties for the analytical elements are illustrated in the figure below.

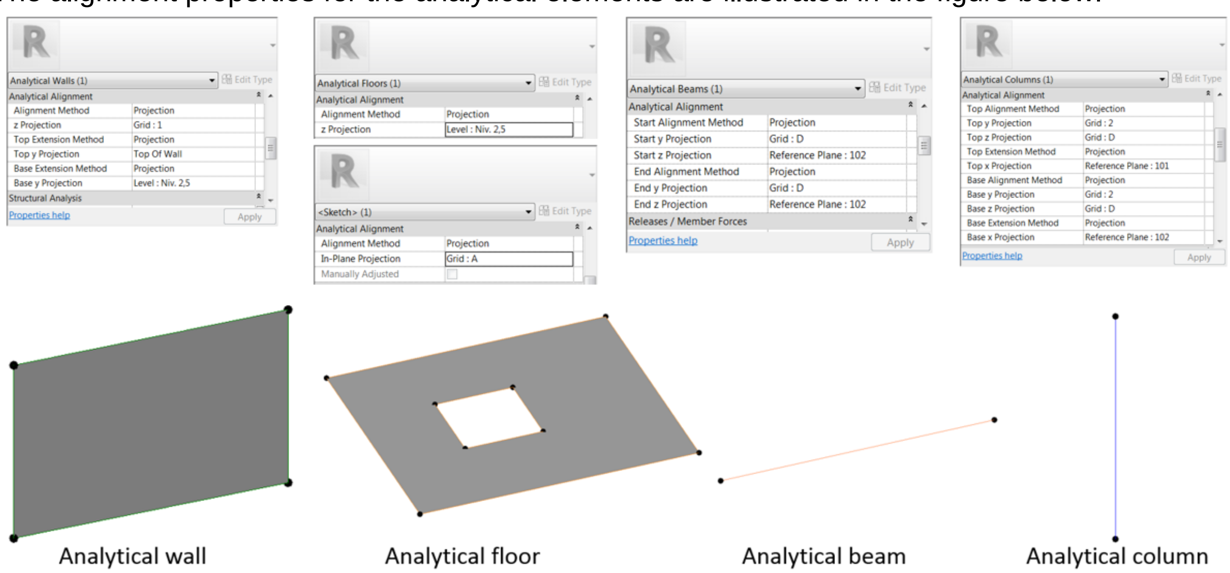


Figure 6: Analytical element properties in Revit

It should be noted that the properties for in-plane alignment of floor elements are found in the floor sketch properties.

When having a building model in Revit, and it is desired to use the analytical model for FEM calculations it is a good idea to introduce an analytical grid system as illustrated in the figure below.

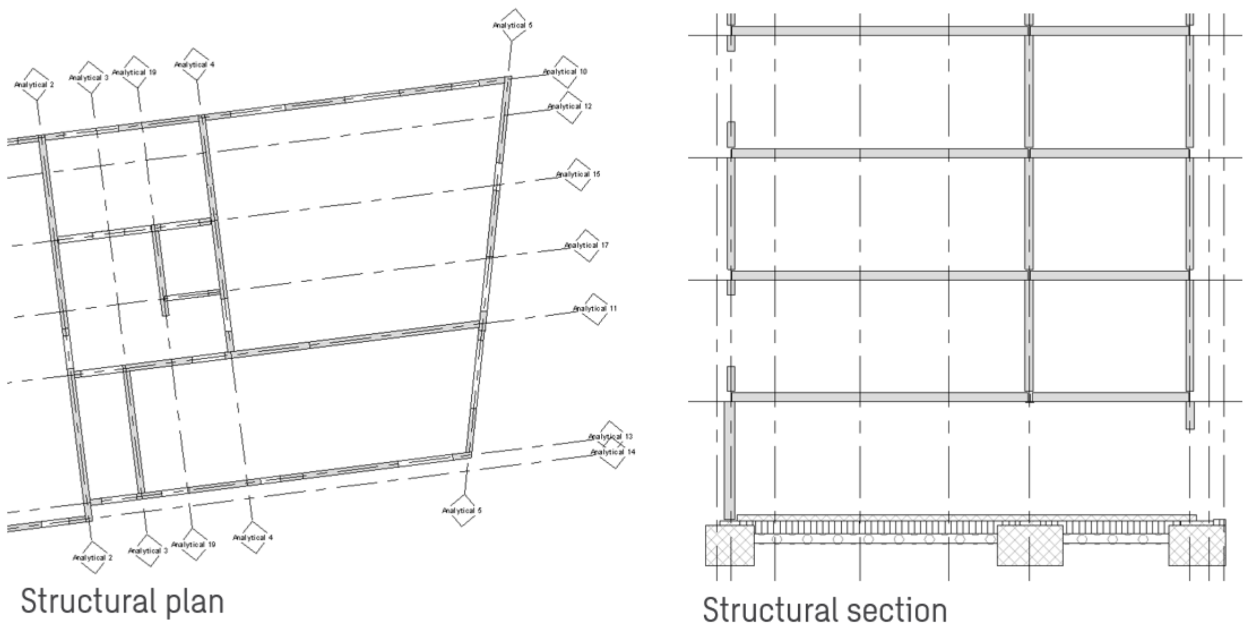


Figure 7: Grid and level alignment

The analytical elements can thereby be locked into the analytical grid system leading to a consistent model. The challenges are now how to fill in the huge amount of properties for all the elements, in order to align the analytical elements to the grid system. This is where Dynamo can be used to automate the process.

Setting alignment properties with Dynamo

Firstly a small Dynamo script is introduced to change the alignment properties from "Auto-Detect" to "Projection". The illustration below shows the part where the alignment properties are changed for wall elements.

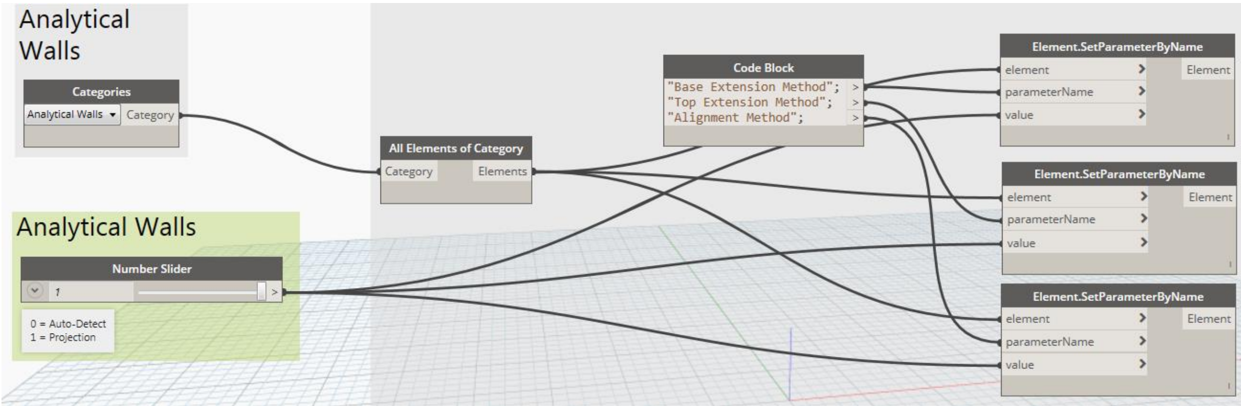
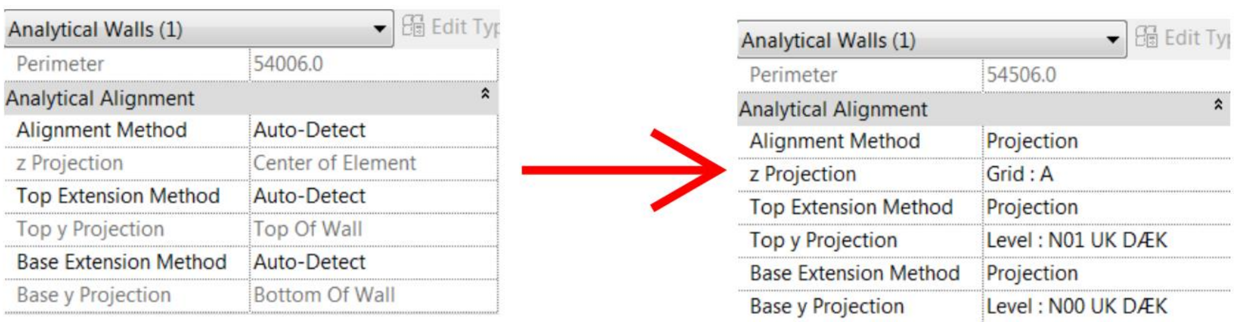


Figure 8: Dynamo script to set projection type

The change in alignment properties are necessary to enable the user defined alignment (see figure below).



Analytical Walls (1)	Edit Typ
Perimeter	54006.0
Analytical Alignment	
Alignment Method	Auto-Detect
z Projection	Center of Element
Top Extension Method	Auto-Detect
Top y Projection	Top Of Wall
Base Extension Method	Auto-Detect
Base y Projection	Bottom Of Wall

Analytical Walls (1)	Edit Ty
Perimeter	54506.0
Analytical Alignment	
Alignment Method	Projection
z Projection	Grid : A
Top Extension Method	Projection
Top y Projection	Level : N01 UK DÆK
Base Extension Method	Projection
Base y Projection	Level : N00 UK DÆK

Figure 9: Change in projection types

Hereafter the alignment properties will be enabled and scripts for correcting elements can be introduced.

The script presented below sets the alignment properties for the wall elements.

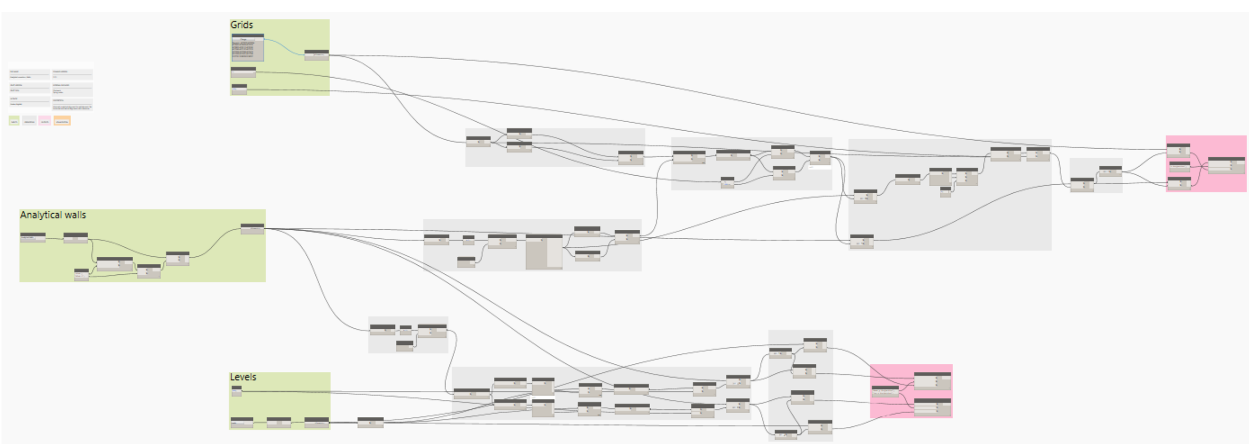


Figure 10: Dynamo script to set alignment for walls

The grid alignments are made by comparing the location line of the wall with the grid line. A user defined tolerance in distance and degrees are introduced to account for small modelling errors.

The node “Vector.AngleWithVector” is used to the parallel check and the node “Geometry.DistanceTo” is used to find the nearest gridline within the tolerance. The setting “Cross Product” is used to compare all gridlines and walls. The parallel check is illustrated below.

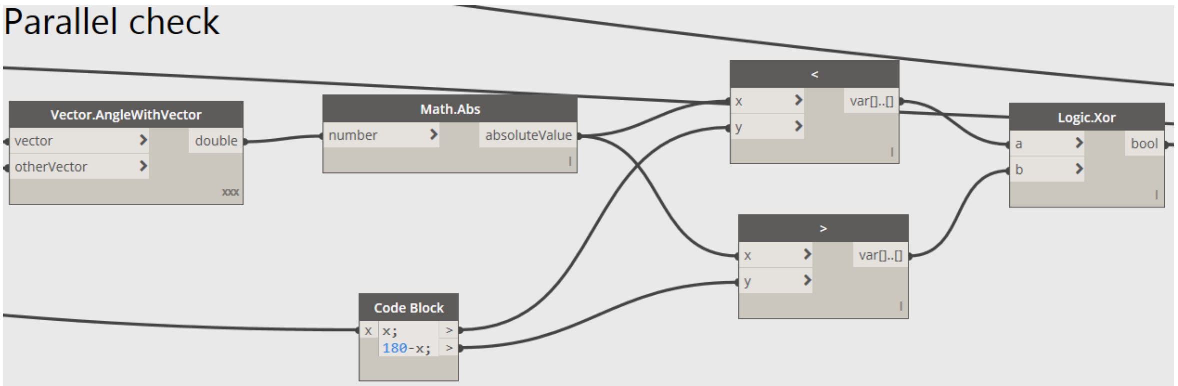


Figure 11: Parallel check in Dynamo

The level alignments are made with use of a bounding box and comparing the z-coordinate max and min point with the z-coordinate of the level.

The alignment properties for the other elements are made with similar methods.

When using Dynamo scripts to fill in the alignment properties consistent FEM models can easily be build and the risk of mesh errors are reduced significantly. An example of a corrected model is illustrated below.

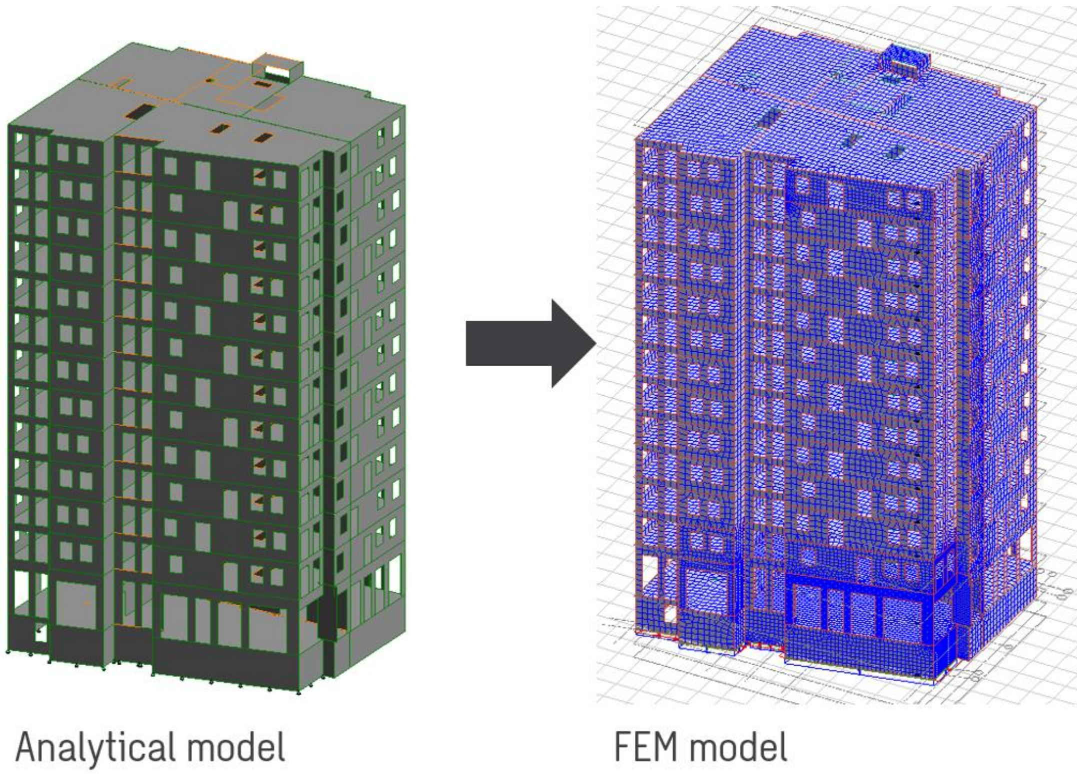


Figure 12: Example of corrected model

Applying loads automatically through Dynamo

When using the analytical model from Revit to structural analysis in FEM programs, it is possible to apply loads to the structure automatically within Revit. This section will explain a procedure to do this.

Load index plans are made in Revit with use of the filter functions. The load index plans explain which dead and live loads there are acting on the structure.

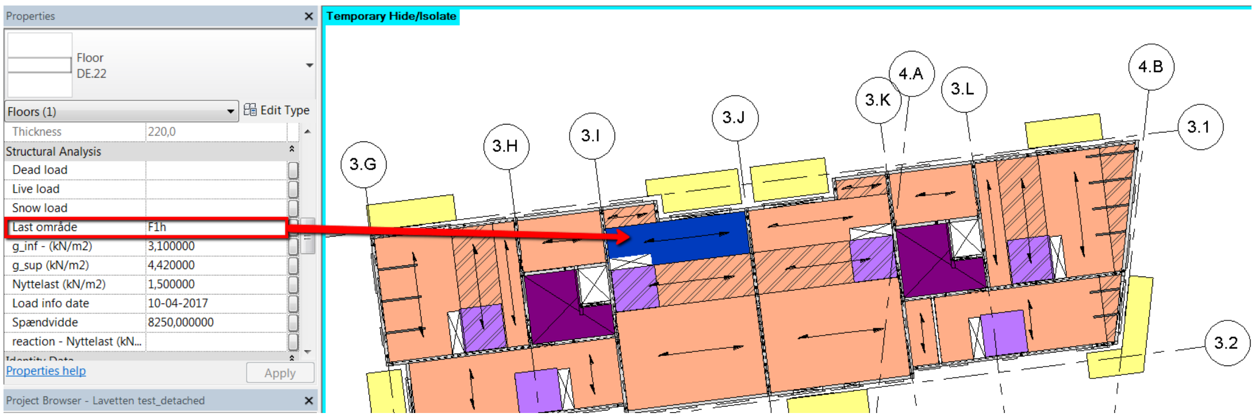


Figure 13: Load index plan in Revit, made with filters

A load tag (could be “F1H”, as shown in the figure above) corresponds to a set of characteristic area loads:

- Minimum dead load
- Maximum dead load
- Live load

The load values that corresponds to the load tags are stored in an Excel spreadsheet, where Dynamo can read the load tag and write the three area loads to parameters for the floor elements. Thereby information about the load acting on the specific floor element are stored in the element properties, and Dynamo can be used to apply the loads automatically to all floor elements.

The Dynamo script to apply hosted area loads are illustrated in the figure below.

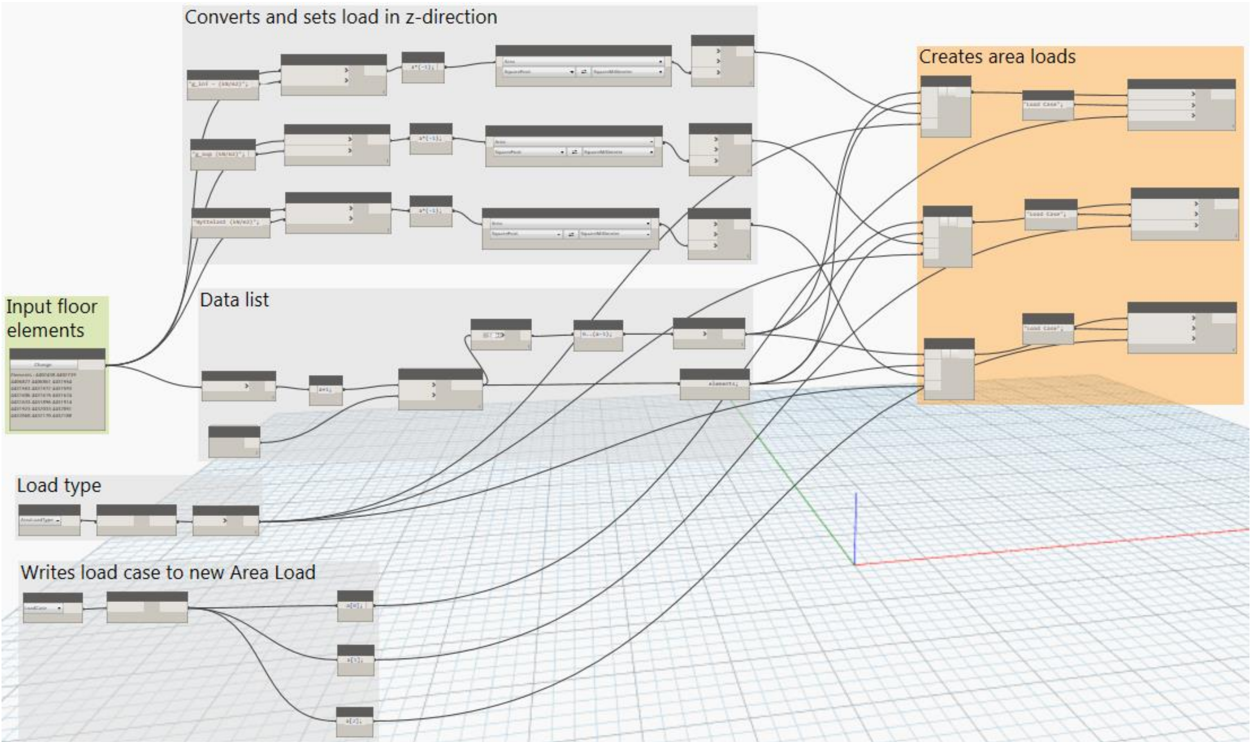


Figure 14: Script to apply hosted area loads

The important part of the Dynamo script is the node with a python script to apply hosted area loads. The python script is shown in the figure below.

```python
import clr
clr.AddReference('ProtoGeometry')
from Autodesk.DesignScript.Geometry import *
# Import tDSType(bool) Extension method
clr.AddReference("RevitNodes")
import Revit
clr.ImportExtensions(Revit.Elements)
# Import geometry conversion extension methods
clr.ImportExtensions(Revit.GeometryConversion)
# Import DocumentManager and TransactionManager
clr.AddReference("RevitServices")
import RevitServices
from RevitServices.Persistence import DocumentManager
from RevitServices.Transactions import TransactionManager
from System.Collections.Generic import *
#Import RevitAPI
clr.AddReference("RevitAPI")
import Autodesk
from Autodesk.Revit.DB import *
from Autodesk.Revit.DB.Structure import *
from System.Collections.Generic import *

doc = DocumentManager.Instance.CurrentDBDocument

data = IN[0]
host = UnwrapElement(IN[1])
F = IN[2]
loadtype=UnwrapElement(IN[3])

loads=[]

for i in data:
        # Start Transaction
    doc = DocumentManager.Instance.CurrentDBDocument
    TransactionManager.Instance.EnsureInTransaction(doc)
    a= AreaLoad.Create(doc,host[i],F[i].ToXyz(),loadtype)
    loads.append(a)

        # End Transaction
TransactionManager.Instance.TransactionTaskDone()

OUT = loads
```

Figure 15: Python script to create area loads

The simple data list used as an input can be made as illustrated below:

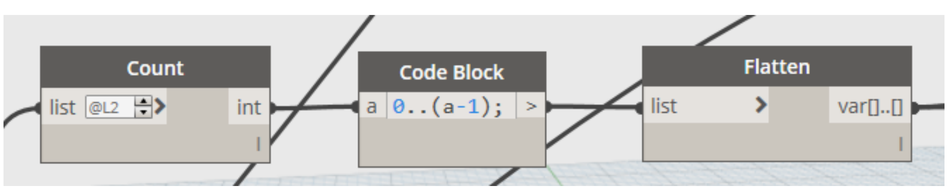


Figure 16: Data list

An example of a corrected model with applied loads are illustrated in the figure below. The loads are created in the correct load cases and will be ready to use in the FEM model.

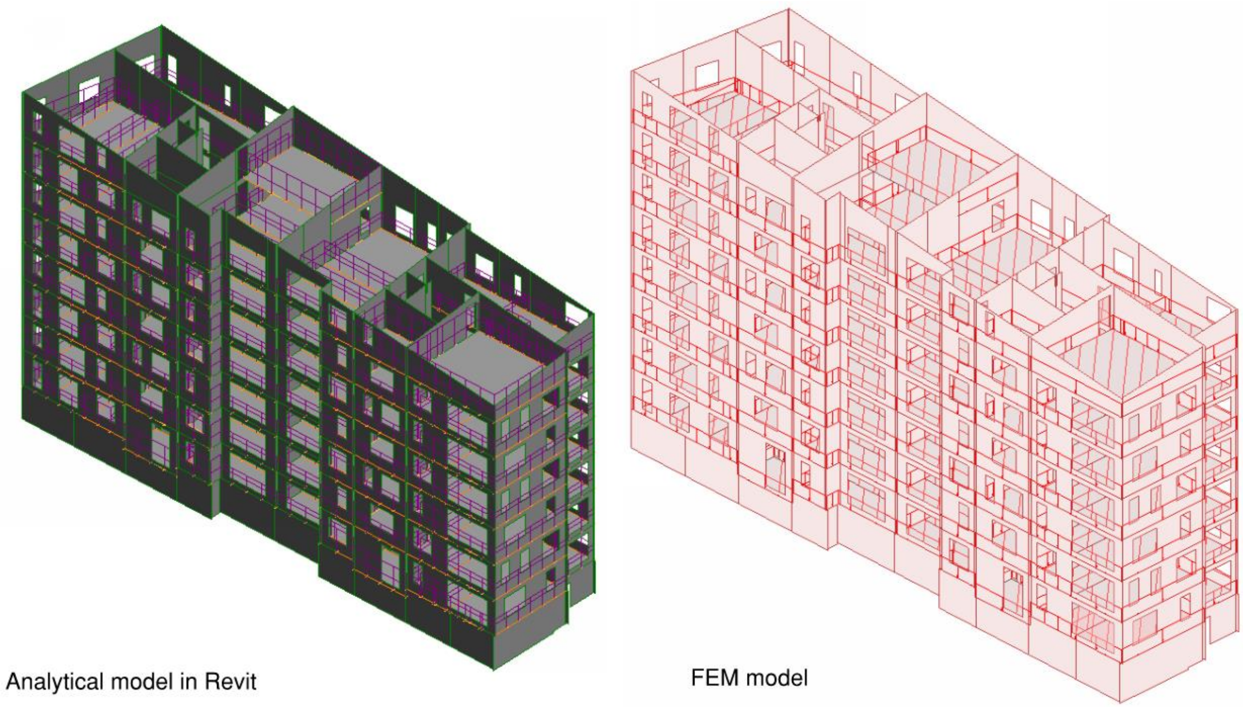


Figure 17: Analytical model with loads in Revit and FEM

Sending results to the Revit elements

When the analytical model from Revit is used, it is possible to write calculation results back to the structural elements. This can be done since the ID's in the FEM model that corresponds with the Revit ID's are known and the connection back thereby is somewhat straight forward. As an example, the results from the FEM model can typically be exported to an Excel spreadsheet whereby a Dynamo script can read the results.

The figure below shows an example where the resulting normal force and moment in the edge connection in the bottom of the wall has been written back to the structural elements for 4 different load combinations. Another possibility could be to write the necessary amount of reinforcement back to the Revit elements. The important thing is, that the models are linked so the user can decide which results should be assessable in the Revit model.

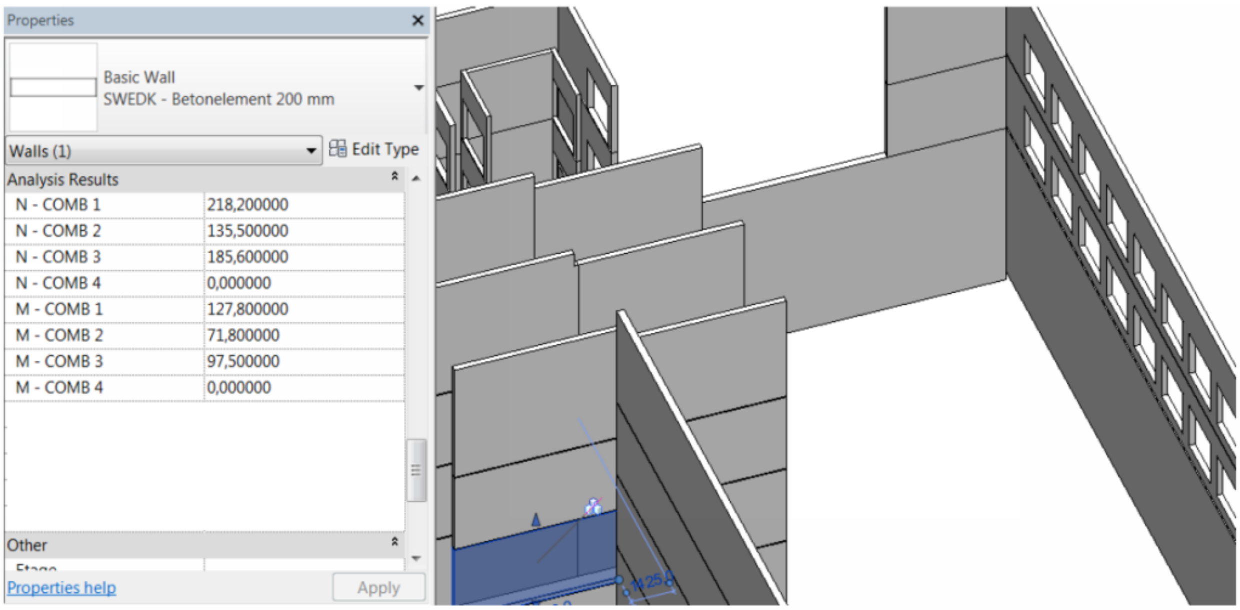


Figure 18: Analysis results in Revit element properties

Conclusion

It can be concluded that a workflow for structural design has been described, with Dynamo as the central tool. Dynamo is used to control the analytical model, apply loads within Revit and send analysis results from the FEM model back to the structural elements in Revit.

An illustration of the workflow in presented in the figure below.

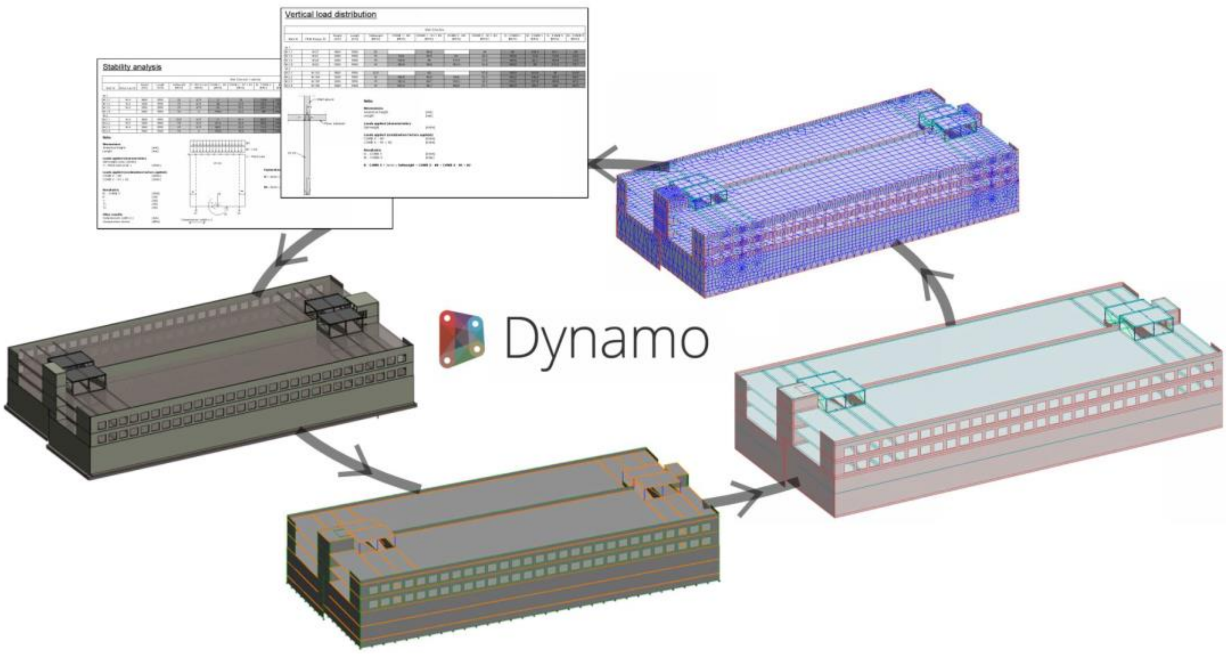


Figure 19: BIM process for structural design