Aalborg University

The effect of interoperability between BIM and FEM tools on structural modeling and analysis.

Master Thesis

Created by
Tamás Drávai
Haroon Khalyar
Gábor Nagy

Supervised by
Kjeld Svidt
Lars Pedersen

June 7, 2016

School of Engineering and Science
Study Board of Civil Engineering
Fibigerstræde 10
9220 Aalborg Øst
Phone: 99 40 84 84
http://www.byggeri.aau.dk

AALBORG UNIVERSITY
STUDENT REPORT

Title: The effect of interoperability between BIM and FEM tools on structural modeling and analysis

Project period: From 01-09-2015 to 08-06-2016

Supervisors: Kjeld Svidt
Lars Pedersen

Number of pages: 115
Delivered: 08-06-2016

Haroon Khalyar | Gábor Nagy | Tamás Drávai

Preface

This project is conducted by a group of students in the 3rd and 4th semester of the Master's Programme in Structural and Civil Engineering at the Faculty of Engineering and Science at Aalborg University. The title of the project is *The effect of interoperability between BIM and FEM tools on structural modeling and analysis* and is composed in the period from 01-09-2015 to 08-06-2016. The aim of the project is to investigate the state of the art Building Information Modeling (BIM) in the Architecture, Engineering and Construction (AEC) industry with a focus on structural modeling and analysis. Kjeld Svidt and Lars Pedersen have supervised this project.
The authors of this project would like to thank for their participation in the interviews and providing valuable support for the project:

- Jens Kristian Lund Birkmose from COWI
- Morten Dalsgaard and Anders Bilgaard from Rambøll
- Mads Vangsgaard from MOE
- Dennis Kristensen and Iwona Budny from StruSoft
- Anders Dalgaard from Trimble Solutions Danmark
- Martin Thaarup and Bjarne Ibsen from ISC Consulting Engineers
- Knud Hjortflod Nielsen from 3D Structural Design

This project contains a main report and two appendices, one external and one internal. The main report focuses on descriptions, approaches and reflections of the findings of this study. The internal appendix contains detailed calculation and overall information that supports the analysis made in the main report but it is not necessary to read it in order to understand the project. The external appendix is given in a CD and contains models made in commercial finite element and Computer Aided Design (CAD) software.
For source reference the Harvard method is used. The reference for books contains the name of the author followed by the year the book was published, edition and publishing. URL references contain title, author, URL and date of download. Figures and tables without a reference is made by the authors of this project. In the bibliography in the end of this project report all references are listed in alphabetical order.
The project is written in past tense when describing work done by project group, while present tense is used when describing how something specific is done generally.

Abstract

This project deals with the effect of interoperability between Computer Aided Design (CAD) and Finite Element Modeling (FEM) software on structural modeling and analysis. Several case studies were conducted, where a building information model was transferred from CAD to FEM software with different data exchange formats. Based on interviews and collaboration with several engineering companies, certain CAD and FEM software were chosen for the case studies. A general structural analysis was made in the FEM software to investigate if the model is imported correctly. The project also deals with data exchange connected to advanced FEA, which includes a non-linear buckling and fatigue analysis.

The conclusion of the case studies is that, data exchange between CAD and FEM software can be useful, but the ease of use depends on both the data exchange format and how the model is created in CAD software. In these case studies, the most successful data exchange was achieved with the direct link between CAD and FEM software compared to open exchange standards. It is also concluded that while there are still technical barriers in front of an efficient data exchange process between CAD and FEM software, especially for open-source solutions, there are multiple ways to make it work fairly successfully. However, in order for a more integrated design approach to be viable in engineering practice, current methods of structural design and communication have to evolve and adapt.

Contents

Chapter 1

Introduction

1.1 CAD assisted structural design

Looking at the general idea of the building design process a crucial part of it is the structural design of the building. This part of the design is responsible for making sure that the structure can withstand the expected loads during its design life with an adequate level of safety. In order to satisfy design criteria the effect of the loads applied to the structure needs to be determined (e.g. displacement, stresses, reaction forces, etc.). In most cases Finite Element Modeling (FEM) is used to achieve this, which requires a simplification of the real building model to create an analytical model with which the calculations can be made. In this project the term analytical model refers to a model which can be used for structural analysis. It does not mean the analysis performed on the model is analytical.

Without including the Computer Aided Design (CAD) model in the structural design process the analytical model is created in the FEM software based on information given by the architect (e.g. drawings, initial dimensions and section sizes, etc.) and the structural engineers judgement and experience (e.g. how to create the analytical model, how should the loads be applied, etc.). Since building design is an iterative process, many changes can occur based on the requirements of any party involved. In a lot of cases these changes affect the load bearing structure of the building which means that the analytical model has to be changed accordingly. If there is no data exchange between the CAD and the FEM software the analytical model has to be changed manually by the structural engineer, which is a time consuming and tedious process. However, in theory, if the data exchange between the two models is implemented the changes made in the CAD model can easily be transferred to the analytical model with a high level of automation. Naturally, the structural engineer can still modify how the change is made in the analytical model. To achieve this, the concept of Building Information Modeling (BIM) can be utilized which is explained in the following section.

1.2 Concept of Building Information Modeling

The following is based on Eastman et al. [2011]. First, a description is given of the current paper-based design and construction methods used by the Architectural, Engineering, and Construction (AEC) industry, which leads to a better understanding of the significant changes BIM introduces.

The current AEC business model is fragmented, and it depends on paper-based modes of communication. Errors in paper documents often cause unanticipated field costs, and delays which often lead to lawsuits between the different parties in a project team. Some efforts have been made to address these issues such as alternative organizational structures; the use of real-time technology e.g. project websites for sharing plans and documents; and the implementation of 3D CAD tools. The timely exchange of information is improved by these methods, but the severity and frequency of conflicts caused by paper documents or their electronic equivalents are not reduced significantly.

A common issue when considering 2D-based communication during the design phase is the time and expense required to generate critical assessment information about a proposed design e.g. cost estimates, energy-use analysis, structural details. These analyses are usually done last in the design process, where it is already too late to make significant changes.

CAD systems generate digital files, that consists of vectors, associated line types and layer identifications. As CAD systems are further developed the users have the possibility to share data associated with a given design. On the other hand, a model of a building created by a BIM tool represents the real physical objects more closely, as, for example a beam, is modeled with a beam object instead of vectors. This also enables the model to contain a magnitude of various types of information about each object such as element type, material properties, structural purpose, manufacturing properties, scheduling, etc. With BIM, the model can also support multiple different views of the data contained within a drawing set, including 2D and 3D views. BIM is defined as a modeling process and associated set of processes to produce, communicate and analyse building models and the vision for it is an improved planning, design, construction, operation and maintenance process. When using BIM, one or more accurate virtual computer models of a construction are created digitally, that contain precise geometry and data needed to support the construction, fabrication, and procurement activities through which the building is realized. This process is illustrated in Figure 1.1.

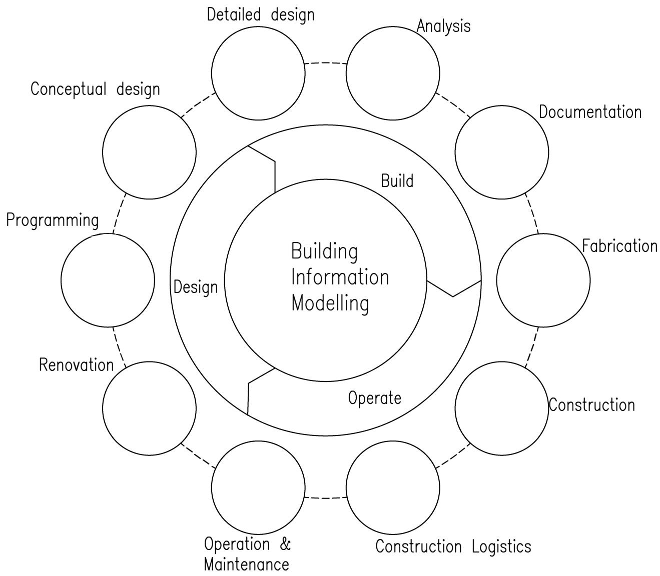


Figure 1.1: Illustration of Building Information Modeling process, based on Advanced Solutions, Inc. [2016].

When this process is adopted well, it facilitates a more integrated design and construction process which results in better quality buildings at lower costs and reduced project durations compared to traditional paper-based design and construction methods. The general relationship between effort and time is illustrated in Figure 1.2.

The time is divided in intervals starting with pre-design - PD, schematic design - SD, design development - DD, construction documentation - CD, procurement - PR, construction administration - CA, and finally operation - OP (this also includes demolition of the building at the end of its life time).

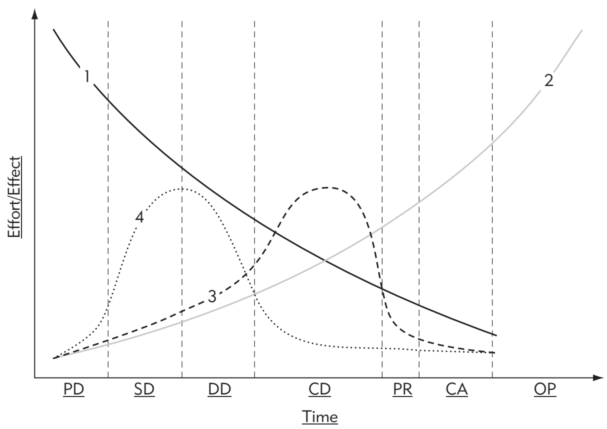


Figure 1.2: Effort as a function of time during a project. [Eastman et al., 2011]

In Figure 1.2 line 1 indicates the ability to impact cost and functional capabilities. Line 2 shows the cost of design changes e.g. it is less expensive to introduce design changes during PD compared to introducing design changes during CD. Line 3 illustrates how effort is disturbed traditionally, while line 4 indicates how it can be redistributed by the use of BIM. BIM has the ability to automate standard forms of detailing and therefore reduces the amount of time required for producing construction documents. The design process using BIM (line 4) aligns effort more closely with the value of decisions made during the design and build process (line 1) and the cost of design changes within the lifetime (line 2), and is therefore the preferred design process.

The concept of parametric objects are crucial in order to understand how BIM is different from traditional 3D objects. Even though a simple line in a CAD software is also a parametric object, the capabilities of parametricity with BIM are more extensive, in addition to modeling the physical object more accurately. Some of the definitions of a parametric object is listed below:

- Consists of geometric definitions and associated data and rules.
- Objects have the ability to link to or to receive, or export a set of attributes e.g. structural materials, to other applications and models.
- A higher possibility to identify if a particular change violates object feasibility regarding size and manufacturability.

Challenges are also expected when using BIM. Since it is a relatively new technology process in the AEC industry it will cause significant changes in the relationships of project participants and the contractual agreements between them. In addition, if the architects create their design using a BIM tool, the model may not have enough detail for use for the design phase, and thus a new model may be created for design purposes. Another challenge can be to export the models created by architects to engineers. They may not use the same modeling tools, which adds complexity to the project. This issue is the main point of inquiry of the master thesis, which is elaborated on in the following.

1.3 Data exchange between CAD and FEM software

This chapter explains the general concept of data exchange between CAD and FEM software. Furthermore, the choice of CAD and FEM software that are used in this project is described.

To represent the engineering landscape of Denmark and by taking into account the available features connected to CAD and FEM interoperability, the following software were chosen for the project after consulting with leading engineering companies such as COWI and Rambøll. For all software the latest available student version was used.

CAD software

- *Autodesk Revit 2016*
- *Tekla Structures Learning 21.1*
- *Bentley AECOsim Building Designer V8i*
- *Autodesk AutoCAD 2015*
- *ProStructures V8i for AutoCAD 2015*

FEM software

- *Autodesk Robot Structural Analysis 2016*
- *FEM-Design 15*
- *Bentley STAAD.Pro V8i*
- *RFEM 5.06*
- *Abaqus/CAE 16.4*

In the report *Autodesk Robot Structural Analysis 2016* is referred to as *RSA*.

1.3.1 Exchange methods

As a cornerstone of the concept of BIM there is a vast array of information that can be input in the CAD model such as structural properties, manufacturing properties, scheduling data, etc. Obviously, there is no need for all of this information to be transferred to a FEM software only the ones that are relevant for structural analysis. The most important one and the most frequently used is the geometry of the structure. This is needed so that the analytical model can be created. It also requires information about the position of the elements so that they can be inserted into the coordinate system of the FEM software in addition to the type of the elements. To perform the calculations the section profile and material properties of each element are needed. Furthermore, if it is available in the CAD model the loads, supports and releases can also be transferred. Although, these are usually better left to the structural engineer to specify in the FEM software. In the following figure the general transferring methods, from CAD to FEM software, relevant to this project can be seen.

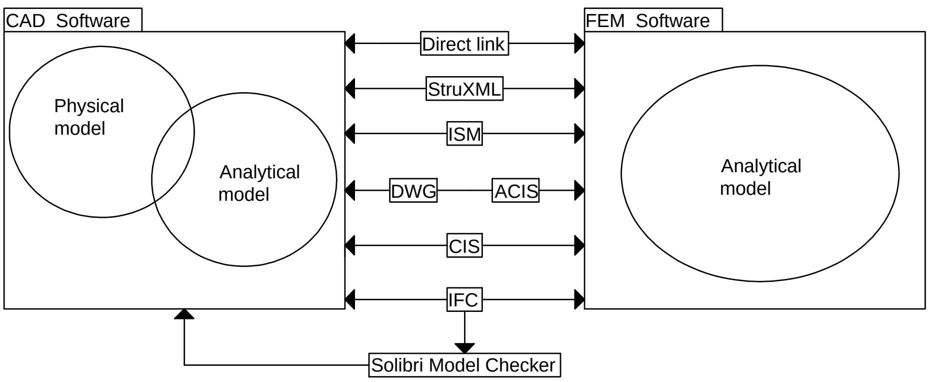


Figure 1.3: General transfer methods.

The arrows on the figure explain the direction of the data exchange. The boxes represent the different formats for the different exchange methods. Moreover, the IFC format can be opened with the *Solibri Model Checker* to detect errors, which is elaborated further in Section 4.4.1 (p. 24).

The specific data exchange methods for the various cases are described in detail in Chapter 4 (p. 19) and Chapter 6 (p. 57).

1.4 Problem statement

The purpose of this Master Thesis is to examine the technical capabilities of the data exchange between CAD and FEM software and investigate if this integrated design approach is applicable in Danish engineering practice.

The goal of the Master Thesis is to find the answers to the following problem formulations:

How can the implementation of BIM in the design process help structural design?

Which method of data exchange has the highest potential and which direction should the field of data exchange be developed?

How can this interoperability be implemented in engineering practice?

Chapter 2

Solution strategy

In this chapter the strategy to find the solution for the problem statements is outlined. Including an explanation of how the problems stated above are investigated and what kind of methods are used to find answers to them.

As the concept of BIM and its place in the building design process is very complex, the focus had to be put on specified aspects of it in order to fit the scope of the thesis. There are two parts of the connection between BIM and FEM tools this thesis is examining. The first one is the technical capabilities of the most widely used commercial CAD and FEM software and file formats in Denmark. The second one is its current level of practical viability at Danish engineering companies including the obstacles in front of a more integrated data exchange and possible ways for future development.

Firstly, a literature review is performed to understand and leverage previous findings of scientific publications and to identify current research trends within the topic of CAD and FEM software interoperability.

To assess the technical capabilities of the link between BIM and FEM tools, several case studies were made with increasing levels of structural complexity, varying materials and different combinations of commercial CAD and FEM software. Based on structural complexity the case studies were divided into two categories, simple cases and advanced cases.

The simple cases category consists of a simply supported beam and a single frame using steel, concrete and timber as a material for the beam and steel and concrete for the frame. The purpose of these cases was to examine what kind of data can be transferred from the CAD software to the FEM software and evaluate the different methods of data exchange in terms of technical capacity and practical potential. The beam and the frame were modelled in each of the CAD software and then transferred to the FEM applications with various modes of data exchange. The general method of each data exchange method is explained first and then the encountered issues are discussed individually for each case. An evaluation of each link is presented at the end of the simple structures section.

The used software and links between them can be seen in Figure 2.1

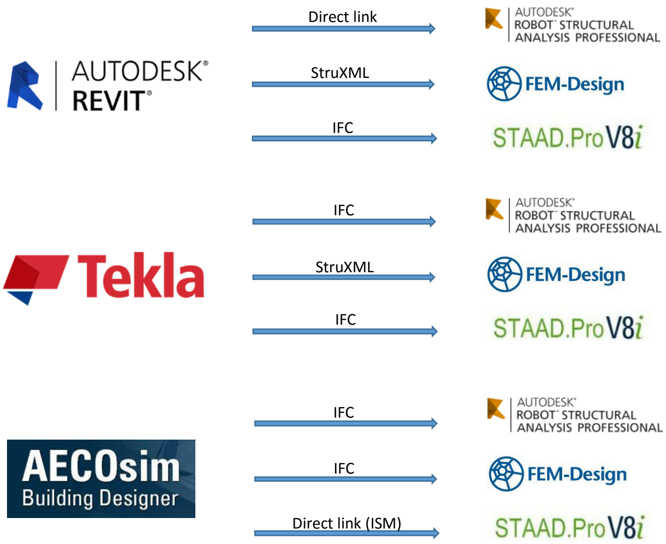


Figure 2.1: Software links for simple cases.

The advanced cases portion of the report consists of two parts. In the first part the CAD model of a platform structure, supplied by an engineering company called ISC and a conveyor bridge, supplied by another engineering company named 3D Structural Design, were transferred to FEM applications and a general structural analysis is performed. The results were then compared to results from the FE model of the structures made by the above mentioned companies. This way the effect of including the CAD model in the design process could be studied. The second part deals with the examination of data exchange for advanced FE analyses made in *Abaqus/CAE*. Two problems were chosen as case studies to examine the benefits and drawbacks of using the CAD model as the basis for model creation. The first one was the determination of the buckling resistance of a frame corner with both a linear and nonlinear analysis. The second one was concerned with fatigue analysis using the Hot Spot & Notch Stress methods. Figure 2.2 shows the data transfer methods used for the advanced cases.

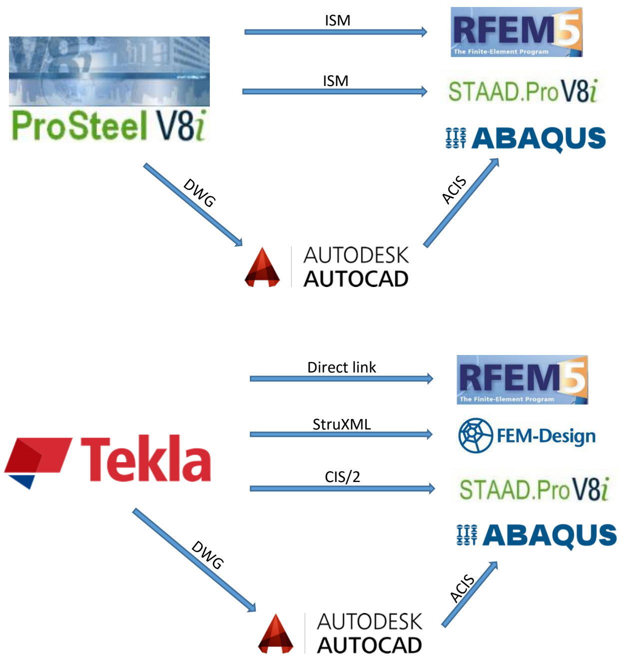


Figure 2.2: Software links for advanced cases.

To examine the practical viability of the different data exchange methods and to get an idea about the actual implementation of BIM and FEM interoperability in engineering practice, several interviews were conducted with structural engineers and BIM professionals from COWI, MOE, Rambøll and 3D Structural Design. Every interview was based on the same predefined set of questions which can be found in Appendix A. The interviews were used to formulate an accurate and up-to-date picture of the real life usage and most pressing issues about the topic at hand. Even though only four interviews were conducted, they were made with four prevalent engineering companies in Denmark, thus the results can be considered representative of the Danish engineering field.

Finally, a conclusion is presented, which reflects the answers to the problem statement that were found during this examination of the subject. Additionally, proposals are presented to identify potential areas of improvement in the BIM and FEM aspects of the building design process that are not only technically possible, but practically viable at the same time.

Chapter 3

Literature review

This chapter presents an overview of the scientific publications and master theses relevant to the topic of data exchange between CAD and FEM software. Although interoperability is a frequently investigated topic, research about data exchange for structural engineering purposes specifically is rather scarce. Nevertheless, there are several noteworthy publications and theses for this topic, which are discussed below.

Tamás Rácz and Thomas Olofsson published an article in 2009 in which they investigated the challenges of interoperability from the standpoint of a structural engineering software vendor, namely StruSoft AB. It highlights that even though standardization requires a lot of work, being able to communicate effectively between parties is a fundamental. It also emphasizes that open solutions like IFC should be used over proprietary ones to a enable free flow of information. [Racz & Olofsson, 2009]

António Grilo and Ricardo Jardim-Goncalves wrote an article in 2010 concerning a value proposition on the interoperability of BIM. They suggest that instead of focusing only on the technological aspects, analyses should be carried out for the value propositions at the business level. The authors concluded that in order for it to succeed changes are needed for the information system, business processes and a new way of managing business relationships. [Grilo & Jardim-Goncalves, 2010]

Qin Ling, Deng Xue-yuan and Liu Xi-la developed a framework for an IFC based central BIM system in 2011. It included a bidirectional interface between BIM and a central XML based FEM that is able to communicate with commercial FEM applications. Through testing the authors showed that the developed system can successfully integrate CAD environments with several commercial FEM software. [Qin et al., 2011]

Oberwinter et al. [2013] published a paper about a research study at the Vienna University of Technology in 2013 concerning an interdisciplinary planning process with BIM support. They examined connections between various combinations of commercial BIM tools and FEM applications. The results showed that BIM integration requires comprehensive standardization and policy development. It was also concluded that open source and proprietary forms of data exchange as well experience difficulties for geometry interpretation and that regardless of the way an architectural model is built up some problems are bound to arise for one or more disciplines. The paper also highlights that one of the fundamental issues in this topic is the agreement between the parties on how detailed the BIM should be and how it should be built.

As a continuation of the research study in the previous paragraph, another paper was published by Kovacic et al. [2014] in 2014. Putting a bigger focus in building physics than before showed that both open platform and proprietary BIM tools result in several problems for data exchange. They also highlight that a proprietary one-platform BIM system can experience severe problems in cases where multiple companies or consultants work on the same project.

Various master theses were also made in this topic. A thesis made by Anne Kathrine Nielsen and Søren Madsen at Aalborg University in 2010 investigated several methods of data connections between multiple CAD and FEM applications. From these case studies they found that at the time of their writing direct links between application gave the best results while the IFC exchange had severe limitations. However, they do emphasize that a general development goal should be for FEM software vendors to integrate the IFC Structural Analysis View which can greatly improve the data exchange for structural engineering purposes. [Nielsen & Madsen, 2010]

In another thesis made by Thomas Hansen at Technical University of Denmark (DTU) in 2011 a standalone programme was developed that was able to transfer data through IFC from *Tekla Structures* and a specific analysis software for beam elements. The study showed that there is a need for the IFC schema to be developed further. However, it also showed that the biggest issues lied with the existing BIM software structure. The author also concludes that in order for the implementation of BIM to be useful it should be utilised in all phases of the design process. [Hansen, 2011]

A further thesis from DTU made by Kenneth Zollfrank Gustavsen in 2012 deals with investigating the level at which an architectural building information model can be used for structural analysis purposes. It examined the data exchange between various commercial BIM and FEM applications using both proprietary and IFC based formats. The tests showed that to use the architectural model for structural analysis it must contain an analytical representation of the structure. Additionally, it also concluded that the direct links between applications performed better in the case studies, however the author recommends the implementation of the IFC Structural Analysis View to ensure that the structural engineer has the freedom of choice when it comes to the choice of which software to use for structural analysis. [Gustavsen, 2012]

The reviewed papers show that the data exchange between CAD and FEM applications experiences many issues. From a technical standpoint a common trend is that proprietary direct links between applications currently perform better than open source solutions. However, every paper emphasises that instead of a closed system an open one should be the focus of development in the future. Several articles also conclude that aside from technical difficulties interdisciplinary communication and project management practices also need to change and adapt to achieve a successful and more integrated data exchange model. These papers also show that this topic is still in need of investigation and that collaboration with the companies involved in the design process is necessary to find solutions that have viability for practical use.

Part I

Simple cases

Chapter 4

Data transfer methods

In this chapter the various data exchange links shown in Figure 2.1 are discussed which are used for the simple cases. The general process is shown in the following and necessary steps are explained for each link in Appendix C.

4.1 Revit - RSA direct link

The exchange process can be seen in Figure 4.1.

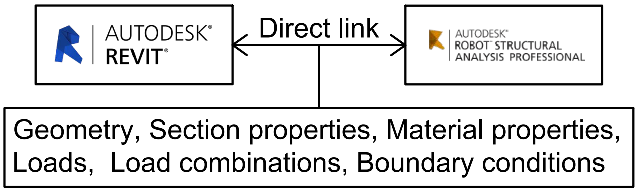


Figure 4.1: Data exchange between *Revit* and *RSA*.

The direct link between *Revit* and *RSA* is a highly capable and easy to use method of data exchange. The *Revit* software provides a physical model of the structure and the corresponding analytical model in addition to many other properties relevant to structural analysis like section profiles, material properties, boundary conditions, etc. If both software are installed on the same computer there will be an additional command in the user interface that allows for specifying and initiating the data transfer. If this is not the case then an intermediate file format (.smxx) can be used for transferring information, however with a few restrictions to the type of information that can be carried over (e.g. steel connections can not be transferred this way). Another advantage of this link is that it is bidirectional, meaning that the *RSA* model can also be transferred back to *Revit*. This is a really important feature because in many cases after the model has been analysed in *RSA* changes need to be made to fulfil design criteria. With the use of the bidirectional link the changes made in *RSA* can easily be reflected in the *Revit* model. Not only can it transfer the changes to *Revit*, but it can also transfer static analysis results from *RSA*. However, this latter option is only available if *Revit* and *RSA* are installed on the same computer. It can also be specified whether the whole *Revit* model should be transferred or only certain parts of it, which is useful for situations where only certain parts or structural systems need to analysed.

A detailed technical description about the way the model elements are transferred can be found in Autodesk [2014].

Before the data transfer could be done the *Revit* model needed to be properly prepared for it. The most important aspect to investigate was the integrity of the analytical model. In order to be able to perform analysis in the FEM software the analytical model needed to be continuous, the nodes and elements appropriately connected, loads and boundary conditions defined. The loads and boundary conditions are an optional feature for the CAD model as based on industry practice these are handled by the structural engineer in the FEM software. However, the analytical model must be prepared correctly in the CAD software, otherwise adjustments need to be made in the FEM software, which can be time consuming and defeats the purpose of the data transfer. Additionally, the elements need to be modeled with the correct type (e.g. the beam was modelled as a structural beam and not an architectural beam) and every information that needed to be transferred had to be input to the model (e.g. material properties were given, boundary conditions were properly set, etc.).

4.2 StruXML

The StruXML format is used to exchange data from *Tekla Structures Learning* to *FEM-Design*, and from *Revit* to *FEM-Design*. The exchange process is shown in Figure 4.2.

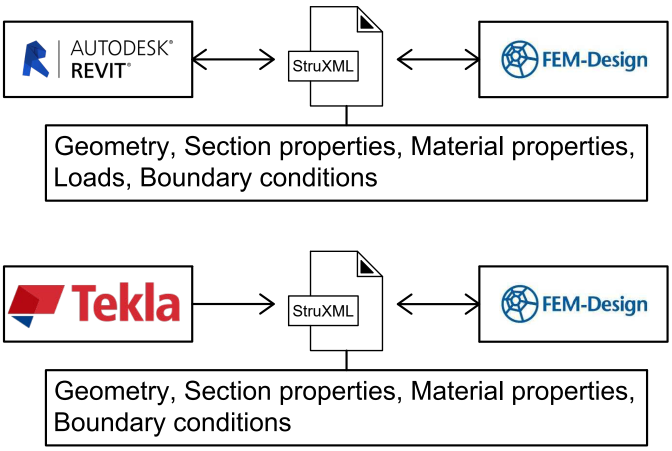


Figure 4.2: Data exchange with StruXML.

The StruXML file format is developed by StruSoft to enable a link between CAD software and *FEM-Design*. It is available for both *Revit* and *Tekla Structures Learning*, however the implementation for the two is different. For *Revit* an add-in is available that allows for a bidirectional link with *FEM-design*. It enables for the transfer of the structural elements, material properties, boundary conditions, geometric eccentricity and loads. The position (center of gravity) of the beam axis (local axis) can be set by the user. For *Tekla Structures Learning*, there is a separate programme to create the StruXML file. Also, this link only works from *Tekla Structures Learning* to *FEM-Design*, there is no data exchange in the other direction at the time of this writing. Furthermore, the *Tekla Structures Learning* export tool does not have the same capabilities as the *Revit* add-in. Loads and beam eccentricity can not be transferred from *Tekla Structures Learning*. What is true for both versions of the StruXML data exchange is that the materials and sections have to be mapped. This means specifying how the the properties from one software should be translated to properties of the other software, by mapping for example the material in the *Revit* library to the same material in the *FEM-Design* library. This process has to be done manually by the user in every case.

The technical aspects are described in detail in StruSoft [2015a] and StruSoft [2015b].

4.3 ISM

Integrated Structural Modelling (ISM) is the direct link between Bentley products. The exchange process is shown in Figure 4.3.

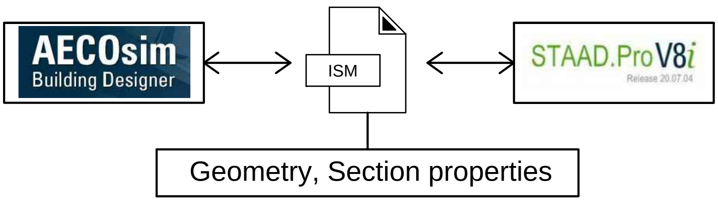


Figure 4.3: Data exchange with ISM.

Although, it is called the direct link, to use this option a third program, called *Structural Synchronizer*, has to be used to achieve the transfer from one application to another. Furthermore, at this time, ISM is only capable of transferring structural model geometry, section and material properties. Loads and boundary conditions can not be transferred with this link.

A detailed technical description can be found at Bentley [2009].

4.4 IFC

The process of data exchange can be seen in Figure 4.4.

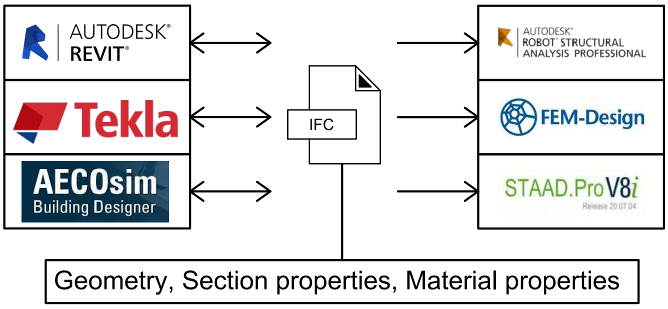


Figure 4.4: Data exchange with IFC.

IFC stands for Industry Foundation Classes and it is both an open standard for exchanging data and a file format that is developed by buildingSMART. It is a non-profit international organisation with an aim of improving the building industry through open, international standards. A fundamental part of this is the development of open shareable asset information, which is achieved by the IFC standard. The main goal of IFC is to make it easier to share information across various software applications to increase productivity and sustainability. To achieve this, a common data schema is developed by buildingSMART which enables the storage and exchange of data between different software applications. In order for a software to work with IFC they need to be certified by buildingSMART. There are approximately 150 different software that support IFC. The IFC standards are constantly developed and as with other products new features are added to it in each release. The current release is IFC 4 which was released in 2013, however it is still not supported by all of the software that is included in the thesis. For this reason the previous release, IFC2x3, was used.

An important part of the IFC standards is the Model View Definition (MVD), which is defined by buildingSMART as "a subset of the IFC schema, that is needed to satisfy one or many Exchange Requirements of the AEC industry." [Model Support Group of the IAI, 2007]

The exchange requirements are defined by the Information Delivery Manual (IDM) which provides specifications about the information needed for a certain role in a project. The following MVDs are currently available as official buildingSMART MVDs for IFC2x3:

- IFC2x3 Coordination View Version 2.0
- IFC2x3 Structural Analysis View
- IFC2x3 Basic FM HandOver View

IFC2x3 Coordination View Version 2.0:

The Coordination View is the most frequently used view of the IFC schema and its purpose is to share information between the architectural, structural and mechanical engineering principles during the design of a building. It holds information about spatial structure, building, and building service elements to allow for the coordination of design between the above mentioned principles. It allows shape representation for all elements with non-parametric shape, and for certain standard elements parametric shapes are supported. [Model Support Group of the IAI, 2007]

IFC2x3 Structural Analysis View:

The Structural Analysis View is suited for transferring a structural analysis model that was created in a CAD software to a structural analysis software. An important feature of it is that it allows for the same structural analysis model to be analysed by various analysis applications. Aside from the structural representation of the analytical model and material and profile information it is capable of exchanging information about loads, load groups and combinations, connections and boundary conditions. [Model Support Group of the IAI, 2007]

IFC2x3 Basic FM HandOver View:

The Basic FM HandOver View is concerned with data exchange to facility and maintenance management applications, including information about spaces, equipment, etc. [Model Support Group of the IAI, 2007]

The level of implementation of the IFC standards is different in each application that was used for the project. Unfortunately, the IFC2x3 Structural Analysis View is not supported by any of them, thus the IFC2x3 Coordination View Version 2.0 was used. This greatly limits the exchange of information that is relevant to structural analysis, as analytical models, loads and boundary conditions can not be transferred with this MVD. Both *Revit* and *Tekla Structures Learning* allow for a high level of customizability when it comes to exporting a model to IFC. With the use of custom property sets even user-defined properties can be exported for both applications. However, this requires the manual mapping of the custom property from the applications own property library to an appropriate IFC entity.

In order to utilize the benefits of data exchange with IFC, as with the other exchange methods, the CAD model needed to be adequately prepared. The concept of this is the same as it was described for the *Revit - RSA* link. As it was mentioned earlier, only *Revit* supports the IFC4 standard. For this reason the IFC2x3 standard was used with the Coordination View 2.0, but the potential improvements of using IFC4 is discussed at the end this section. The method to export the CAD model to the IFC file was different for each software so it is described separately in Appendix C.

4.4.1 Solibri Model Checker

To properly assess the capabilities of the IFC standard itself and its implementation in the different software, an intermediate step was included in the data exchange process. This step was to examine the exported IFC file from the CAD applications in an independent software designed to manage, organize and verify IFC files called *Solibri Model Checker*. With its help it was possible to check if the IFC file created by a CAD application contained every information that was required for it and in the correct data structure. If any issues arose during the data exchange process this step made it possible to locate the source of the problem. For example, if a specific information was missing from the FEM software after import of the IFC file, but it was present in the correct form in *Solibri Model Checker*, then the issue must be with the importing process as the IFC file itself was verified beforehand. *Solibri Model Checker* also has additional features that make it a powerful tool for clash detection, workflow organization and communication between different parties of the design process. However, these aspects of it were not used in this project. The model of a steel frame opened in *Solibri Model Checker* can be seen in Figure 4.5.

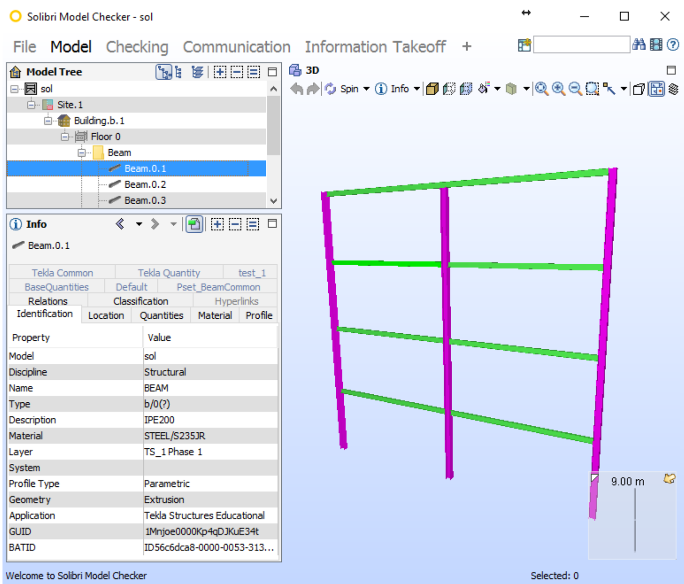


Figure 4.5: Model of a steel frame in *Solibri Model Checker*.

4.4.2 Potential improvements for IFC

The following two sections describe potential improvements, regarding the use of IFC, that have the possibility to be currently implemented. However, for various reasons they are not implemented or not widely used yet.

Structural Analysis View

Regarding structural analysis, meaningful improvements can be achieved by using the Structural Analysis View instead of the Coordination View 2.0, because in addition to the transfer of the analytical model, it would also allow for the transfer of loads, load groups and combinations, connections, boundary conditions, and of course material and profile data. Beside loads themselves displacements can also be defined as structural actions. Furthermore, relationships between structural actions and reactions can be established, so that for example a support reaction in one analytical model can be taken up as a load in another connected system. An example for the grouping of different loads based on their types can be seen in Figure 4.6.

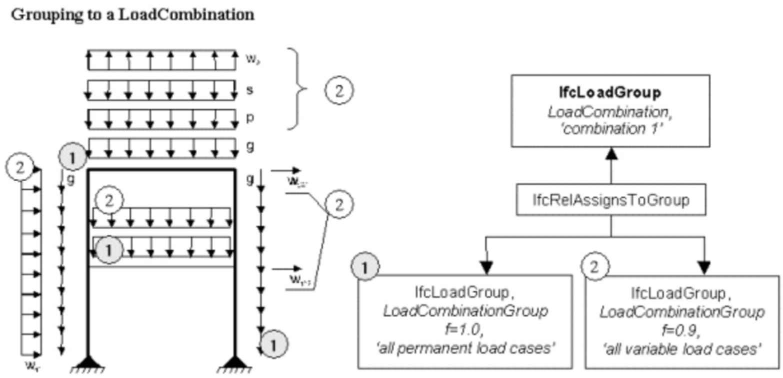


Figure 4.6: Example for defining load groups. [Model Support Group of the IAI, 2007]

It is evident from the above mentioned facts, that the Structural Analysis View is a significant improvement over the Coordination View 2.0 with regards to structural analysis and has a potential to provide benefits for this discipline. However, at the time of this writing none of the software included in the project support Structural Analysis View, even though it was released in 2008. Considering the very slow adoption rate of new technologies in the AEC industry, which in many cases can be measured in decades instead of years, this MVD might still be implemented in CAD and FEM software and utilized in the building design process.

IFC 4

The IFC 4 standard is supported by *Revit*, but *Tekla Structures Learning*, *AECOsim*, *RSA*, *FEM-Design* and *STAAD.Pro* do not support it so it could not be used for the project. Nonetheless the potential benefits of it are examined. IFC4 brings many improvements such as general enhances and stability of the IFC specification, new BIM workflows (e.g. 4D and 5D models), more efficient data structure, etc. The full list of major developments can be seen in Liebich [2013]. Regarding structural design, the most relevant improvements are the following:

- Material profile association can be defined for structural steel and timber objects.
- Structural steel and timber objects can be aligned at cardinal points in their cross sections.
- Allows for the use of anisotropic materials.
- Gives better support for detailing by allowing simplified multiple placements (e.g. rebars, fasteners).

Currently, there are two MVDs for IFC4 released by buildingSMART. The IFC4 Reference view is suitable for transferring a reference model to other applications and disciplines. The model can not be modified so it acts as a read-only model. It is useful for clash detection and for tasks that are viewer based and focus only on visualization. The IFC4 Design Transfer View's purpose is to support and enable more efficient coordination of workflows where changes need to be made between different applications and disciplines. For this reason the transferred model is modifiable and the software must be able to preserve higher-level design parameters so that the geometry of interconnected elements are consistent based on these parameters. This MVD can be used for communication and design coordination between an architect and a structural engineer. It should be noted that at the moment none of the MVDs released for IFC4 include entities for loads or boundary conditions. However, it is reasonable to assume that it will be released in the future. But first the FEM applications need to support the IFC 4 standards to make it viable to use for structural analysis.

Chapter 5

Case studies

In this chapter the case studies for the simple structures will be presented that were used to assess the technical capabilities of each software connection. Two simple structures will be examined to explore the basics of the information exchange and identify the links that have the most potential.

5.1 Simple Beam

5.1.1 Model description

To investigate the capabilities of the links the first model was chosen to be a simply supported beam with a span of 5 m. The physical model of the beam is shown in Figure 5.1, modeled in *Revit*. The CAD model includes information about structural type, material and section properties, boundary conditions, loads and load combinations.

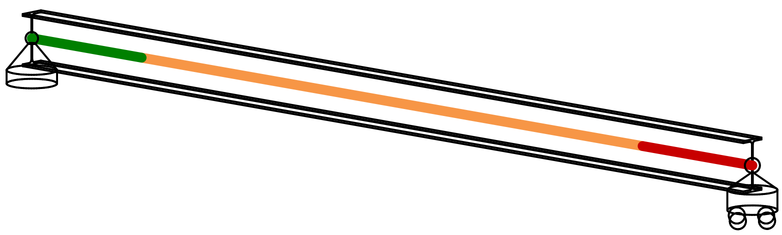


Figure 5.1: Physical model of the beam modeled in *Revit*.

The used cross-sections can be seen in Figure 5.2. The fiber direction of the timber beam is parallel to the longitudinal axis of the beam. The concrete beam was modeled with reinforcement in the CAD applications.

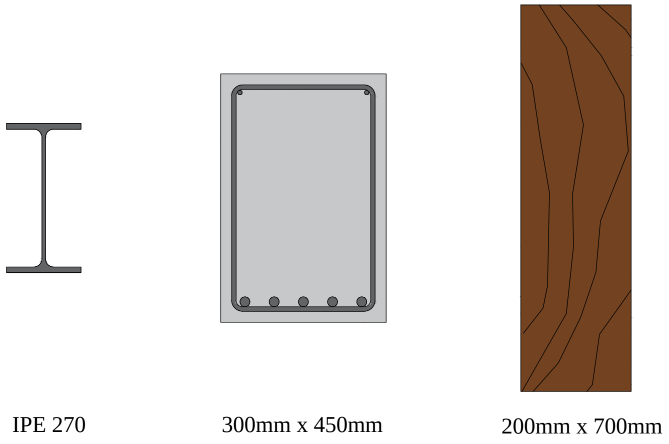


Figure 5.2: Used cross-sections.

Three types of materials were used for the tests to see how they affect the capabilities of the links. A steel, concrete and timber beam was created with the material specifications listed in Table 5.1. Only linear elastic material models were used, since the plasticity phenomena was not relevant to investigate for this topic.

Table 5.1: Material grades.

Material	Grade
Steel	S235
Concrete	C25
Timber	C30

The specific properties modeled in the CAD applications can be seen for each material in Table 5.2, Table 5.3 and Table 5.4.

- RVT: *Revit*.
- TEK: *Tekla Structures Learning*.
- ABD: *AECOsim*.

Table 5.2: Results of the case study for the steel beam.

	RVT	TEK	ABD
Section properties			
Profile name	✓	✓	✓
Height	✓	✓	✓
Width	✓	✓	✓
Web thickness	✓	✓	✓
Flange thickness	✓	✓	✓
Radius	✓	✓	✓
Nominal weight	✓	✓	✗
Area	✓	✓	✓
Shear area	✗*	✗	✗
Moment of inertia	✗*	✓	✗
Torsion constant	✗*	✗*	✗
Warping constant	✗*	✗*	✗
Elastic modulus	✗*	✗	✗
Plastic modulus	✗*	✗*	✗
Radius of gyration	✗*	✓	✗
Geometry			
Length	✓	✓	✓
Position of analytical line	✓	✓	✓
Material properties			
Yield stress	✓	✓	✗
Tensile stress	✓	✓	✗
Young's modulus	✓	✓	✗
Shear modulus	✓	✗	✗
Density	✓	✓	✗
Poisson's ratio	✓	✓	✗
Loads			
Self-weight	✗	✗	✗
Concentrated force	✓	✓	✗
Distributed force	✓	✓	✗
Load combinations	✓	✓	✗
Boundary conditions			
Releases	✓	✓	✗
Supports	✓	✓	✗
Structural member type	✓	✓	✗

- ✓: Possible to define and export.
- ✗: Not possible to define.
- ✗* : Value of the property shown as zero.

Table 5.3: Results of the case study for the concrete beam.

	RVT	TEK	ABD
Section properties			
Profile name	✓	✓	✓
Height	✓	✓	✓
Width	✓	✓	✓
Area	✗+	✓	✓
Moment of inertia	✗+	✗	✓
Effective height	✗	✗	✗
Reinforcement number	✓	✓	✓
Reinforcement position	✓	✓	✓
Reinforcement shape	✓	✓	✓
Reinforcement diameter	✓	✓	✓
Concrete cover	✓	✓	✓
Geometry			
Length	✓	✓	✓
Position of analytical line	✓	✓	✓
Material properties - Concrete			
Characteristic cylinder strength	✓	✗	✗
Characteristic cube strength	✗+	✗	✗
Mean tensile strength	✓	✗	✗
Secant modulus of elasticity	✓	✗	✓
Poisson's ratio	✓	✓	✓
Exposure class	✓	✗	✓
Material properties - Reinforcement			
Yield strength	✓	✗	✗
Class	✓	✗	✗
Young's modulus	✓	✗	✗
Poisson's ratio	✓	✗	✗
Characteristic strain at maximum force	✗+	✗	✗
Density	✓	✗	✗
Loads			
Self-weight	✗	✗	✗
Concentrated force	✓	✓	✗
Distributed force	✓	✓	✗
Load combinations	✓	✓	✗
Boundary conditions			
Releases	✓	✓	✓
Supports	✓	✓	✓
Structural member type	✓	✓	✓

- ✓: Possible to define and export.
- ✓+ : Limited number of cross-sections available.
- ✗: Not possible to define.
- ✗+: Value of the property shown as zero.

Table 5.4: Results of the case study for the timber beam.

	RVT	TEK	ABD
Section properties			
Profile name	✓	✓	✓
Timber type	✓	✓	✓
Height	✓	✓	✓°
Width	✓	✓	✓°
Area	✗°	✓	✓°
Moment of inertia	✗	✗	✓°
Geometry			
Length	✓	✓	✓
Position of analytical line	✓	✓	✓
Material properties			
Bending strength	✓	✗	✗
Tension strength - parallel to grain	✓	✗	✗
Tension strength - perpendicular to grain	✓	✗	✗
Compression strength - parallel to grain	✓	✗	✗
Compression strength - perpendicular to grain	✓	✗	✗
Shear strength	✓	✗	✗
Mean modulus of elasticity - parallel to grain	✓	✗	✗
5 % modulus of elasticity - parallel to grain	✗	✗	✗
Mean modulus of elasticity - perpendicular to grain	✓	✗	✗
Mean shear modulus	✓	✗	✗
Density	✓	✓	✗
Tree species	✓	✗	✗
Loads			
Self-weight	✗	✗	✗
Concentrated force	✓	✓	✗
Distributed force	✓	✓	✗
Load combinations	✓	✓	✗
Boundary conditions			
Releases	✓	✓	✗
Supports	✓	✓	✗
Structural member type	✓	✓	✗

- ✓: Possible to define and export.
- ✗: Not possible to define.
- ✗° : Value of the property shown as zero.

Regarding the transfer to the FEM applications there was an idea of how the transferred analytical model should be. The expected analytical model of the beam can be seen in Figure 5.3.

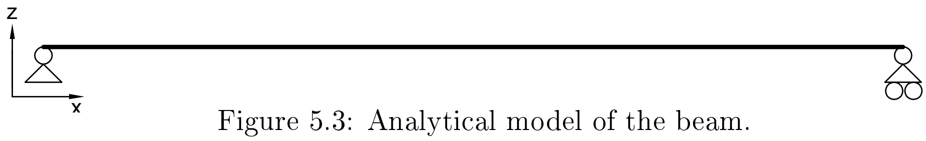


Figure 5.3: Analytical model of the beam.

The loading was specified so that it includes distributed and also point loads. Additionally, the self-weight of each beam is taken into account. The load set-up and their values can be seen in Figure 5.4 and Table 5.5.

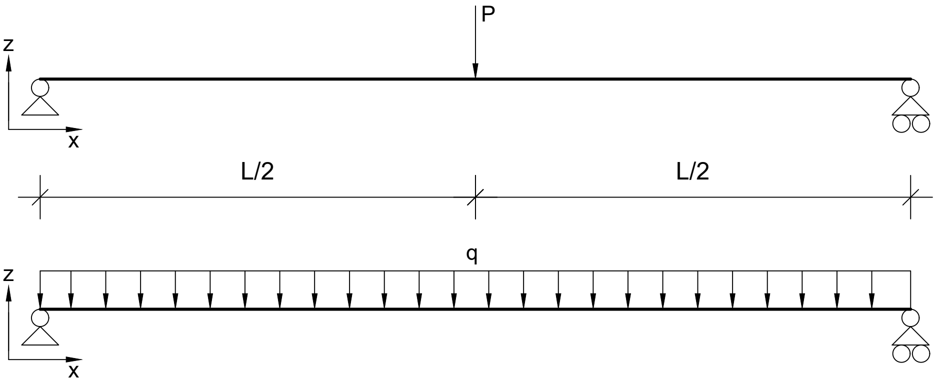


Figure 5.4: Load set-up.

Table 5.5: Load values.

Load	Value
Dead load - P	3 kN
Live load - q	5 kN/m

For the calculation a manual load combination was made according to Eurocode 0. As part of the evaluation of the case studies it was studied if the transferred model was representing the model shown in Figure 5.3. If this was not the case then modifications were made in the FEM software to end up with such a model.

Concerning the structural analysis, the specific tests were the following. The finite element model was used to calculate displacements, moments, shear forces, and stresses. Furthermore, the built-in code check functions were used to perform code checks according to the related Eurocode. This included the calculation of the cross-sectional and buckling resistance and for the reinforced concrete beam the design of the reinforcement. The parameters needed to perform the code checks were defined in the FEM applications in all cases as it was not possible to define them in the CAD software and also no need to transfer them because these should be handled by the structural engineer.

All three models were created in all three CAD applications then were exported using the links described in Chapter 4 (p. 19). The results of the data exchange are presented in Table 5.6, Table 5.7 and Table 5.8 and they will be discussed in the following sections.

Table 5.6: Results of the case study for the steel beam.

	RVT ↓ RSA	RVT ↓ FEMD	RVT ↓ STP	TEK ↓ RSA	TEK ↓ FEMD	TEK ↓ STP	ABD ↓ RSA	ABD ↓ FEMD	ADB ↓ STP
Section properties									
Profile name	✓	✓	✓	×	✓	✓	×	×	✓
Height	✓	✓**	✓	×	✓**	✓	×	✓**	✓
Width	✓	✓**	✓	×	✓**	✓	×	✓**	✓
Web thickness	×*	✓**	✓	×	✓**	✓	×	✓**	✓
Flange thickness	×*	✓**	✓	×	✓**	✓	×	✓**	✓
Radius	×*	✓**	✓	×	✓**	✓	×	✓**	✓
Nominal weight	✓	✓**	×	×	✓**	×	×	✓**	✓
Area	✓	✓	✓	×	✓	✓	×	✓*	✓
Shear area	✓	×	×	×	×	×	×	×	×
Moment of inertia	✓	✓	✓	×	✓	✓	×	✓*	✓
Torsion constant	✓	✓	×	×	✓	×	×	✓*	×
Warping constant	×*	✓	×*	×	✓	×*	×	✓*	×
Elastic modulus	×*	×	×*	×	×	×*	×	×	✓
Plastic modulus	×*	×	×*	×	×	×*	×	×	✓
Radius of gyration	×*	✓	×	×	✓	×	×	✓*	✓
Geometry									
Length	✓	✓	✓	✓	✓	✓	✓	✓	×
Position of analytical line	✓	✓	✓	×	×	✓	×	×	×
Material properties									
Yield stress	✓	✓	×*	×	✓	×*	×	✓	×
Tensile stress	×*	✓	×*	×	✓	×*	×	✓	×
Young's modulus	✓	✓	×*	×	✓	×*	×	✓	×
Shear modulus	✓	✓	×*	×	✓	×*	×	✓	×
Density	✓	✓	×*	×	✓	×*	×	✓	×
Poisson's ratio	✓	✓	×*	×	✓	×*	×	✓	×
Loads									
Self-weight	×	×	×	×	×	×	×	×	×
Concentrated force	✓	✓	×	×	×	×	×	×	×
Distributed force	✓	✓	×	×	×	×	×	×	×
Load combinations	✓	×	×	×	×	×	×	×	×
Boundary conditions									
Releases	✓	✓	×	×	×	×	×	×	×
Supports	✓	✓	×	×	×	×	×	×	×
Structural member type	✓	✓	×	✓	×	×	✓	✓	×

- RVT: *Revit*.
- TEK: *Tekla Structures Learning*.
- ABD: *AECOsim*.
- RSA: *Robot Structural Analysis*.
- FEMD: *FEM-Design*.
- STP: *STAAD.Pro*.

- ✓: Property imported correctly.
- ✓*: Negligible difference between the value in CAD and FEM software.
- ✓**: Property imported correctly, available after design.
- ×: Property not imported correctly.
- ×*: Property value shown as zero.

Table 5.7: Results of the case study for the concrete beam.

	RVT → RSA	RVT → FEMD	RVT → STP	TEK → RSA	TEK → FEMD	TEK → STP	ABD → RSA	ABD → FEMD	ADB → STP
Section properties									
Profile name	✓	✓	✓	✓	✓	✓	✗	✗	✓
Height	✓	✓+	✓	✓	✓+	✓	✗	✓	✗
Width	✓	✓+	✓	✓	✓+	✓	✓	✓	✓
Area	✓	✓	✓	✓	✓	✓	✓	✓	✗
Moment of inertia	✓	✓	✓	✓	✓	✓	✓	✓	✗
Effective height	✗	✗	✗	✗	✗	✗	✗	✗	✗
Reinforcement number	✗+	✗	✗	✗	✗	✗	✗	✗	✗
Reinforcement position	✗+	✗	✗	✗	✗	✗	✗	✗	✗
Reinforcement shape	✗+	✗	✗	✗	✗	✗	✗	✗	✗
Reinforcement diameter	✗+	✗	✗	✗	✗	✗	✗	✗	✗
Concrete cover	✗+	✗	✗	✗	✗	✗	✗	✗	✗
Geometry									
Length	✓	✓	✓	✓	✓	✓	✓	✓	✓
Position of analytical line	✓	✓	✓	✗	✗	✓	✗	✗	✗
Material properties - Concrete									
Characteristic cylinder strength	✓	✓	✗+	✓	✓	✗+	✓	✗	✗
Characteristic cube strength	✓	✗	✗+	✗+	✗	✗+	✗+	✗	✗
Mean tensile strength	✗+	✓	✗+	✗+	✓	✗+	✗+	✗	✗
Secant modulus of elasticity	✓	✓	✗+	✓	✓	✗+	✓	✗	✗
Poisson's ratio	✓	✓	✗+	✓	✓	✗+	✓	✗	✗
Exposure class	✓	✗	✗+	✗	✗	✗+	✗	✗	✗
Material properties - Reinforcement									
Yield strength	✗	✗	✗+	✗	✗	✗+	✗	✗	✗
Class	✗	✗	✗+	✗	✗	✗+	✗	✗	✗
Young's modulus	✗	✗	✗+	✗	✗	✗+	✗	✗	✗
Poisson's ratio	✗	✗	✗+	✗	✗	✗+	✗	✗	✗
Characteristic strain at maximum force	✗	✗	✗+	✗	✗	✗+	✗	✗	✗
Density	✗	✗	✗+	✗	✗	✗+	✗	✗	✗
Loads									
Self-weight	✗	✗	✓+	✗	✗	✓+	✗	✗	✗
Concentrated force	✓	✓	✓+	✗	✗	✓+	✗	✗	✗
Distributed force	✓	✓	✓+	✗	✗	✓+	✗	✗	✗
Load combinations	✓	✗	✓+	✗	✗	✓+	✗	✗	✗
Boundary conditions									
Releases	✓	✓	✓+	✗	✗	✓+	✗	✗	✗
Supports	✓	✓	✓+	✗	✗	✓+	✗	✗	✗
Structural member type	✓	✓	✓+	✗++	✗	✓+	✗++	✗	✗

- ✓: Property imported correctly.
- ✓+: Property imported correctly, available after design.
- ✗: Property not imported correctly.
- ✗+: Property value shown as zero.
- ✗++: Member type changed during the process.

- RVT: *Revit.*
- TEK: *Tekla Structures Learning.*
- ABD: *AECOsim.*
- RSA: *Robot Structural Analysis.*
- FEMD: *FEM-Design.*
- STP: *STAAD.Pro.*

Table 5.8: Results of the case study for the timber beam.

	RVT ↓ RSA	RVT ↓ FEMD	RVT ↓ STP	TEK ↓ RSA	TEK ↓ FEMD	TEK ↓ STP	ABD ↓ RSA	ABD ↓ FEMD	ADB ↓ STP
Section properties									
Profile name	✓	✓	✓	✓	×	✓	-	-	-
Timber type	✓	×	×°°°°	×	×	×°°°°	-	-	-
Height	✓	✓	✓	✓	✓	✓	-	-	-
Width	✓	✓	✓	✓	✓	✓	-	-	-
Area	✓	✓	✓	✓	✓	✓	-	-	-
Moment of inertia	✓	✓	✓	✓	✓	✓	-	-	-
Geometry									
Length	✓	✓	✓	✓	✓	✓	-	-	-
Position of analytical line	✓	✓	✓	×	×	✓	-	-	-
Material properties									
Bending strength	✓	✓	×	×°°°	✓	×	-	-	-
Tension strength - parallel to grain	×°°	✓	×	×°°°	✓	×	-	-	-
Tension strength - perpendicular to grain	×°°	✓	×	×°°°	✓	×	-	-	-
Compression strength - parallel to grain	✓	✓	×	×°°°	✓	×	-	-	-
Compression strength - perpendicular to grain	✓	✓	×	×°°°	✓	×	-	-	-
Shear strength	✓	✓	×	×°°°	✓	×	-	-	-
Mean modulus of elasticity - parallel to grain	✓	✓	×	×°°°	✓	×	-	-	-
5 percent modulus of elasticity - parallel to grain	×	✓	×	×°°°	✓	×	-	-	-
Mean modulus of elasticity - perpendicular to grain	×	✓	×	×°°°	✓	×	-	-	-
Mean shear modulus	✓	✓	×	×°°°	✓	×	-	-	-
Density	✓	✓	×	×°°°	✓	×	-	-	-
Tree species	×	×	×	×°°°	×	×	-	-	-
Loads									
Self-weight	×	×	×	×	×	×	-	-	-
Concentrated force	✓	✓	×	×	×	×	-	-	-
Distributed force	✓	✓	×	×	×	×	-	-	-
Load combinations	✓	✓	×	×	×	×	-	-	-
Boundary conditions									
Releases	✓	✓	×	×	×	×	-	-	-
Supports	✓	✓	×	×	×	×	-	-	-
Structural member type	✓	✓	×	×°°	×	×	-	-	-

- RVT: *Revit*.
- TEK: *Tekla Structures Learning*.
- ABD: *AECOsim*.
- RSA: *Robot Structural Analysis*.
- FEMD: *FEM-Design*.
- STP: *STAAD.Pro*.

- ✓: Property imported correctly.
- ×: Property not imported correctly.
- ×°°: Wrong information imported.
- ×°°°: Information was missing from source file.
- ×°°°°: Material recognised as steel.

5.1.2 Revit to RSA direct link

Model transfer and encountered issues

As it was mentioned earlier this direct link is very capable and well developed, which is also shown by being able to transfer every data that was relevant to structural design from *Revit* to *RSA*. The different types of data that was transferred can be seen together with the other exchange methods in Table 5.6, Table 5.7 and Table 5.8.

For the steel model there were no issues encountered during the data exchange process. This was an indication of the high quality of this link, but it was also due to the fact that this simply supported beam was a very simple model. However, for the timber and the concrete ones some problems arose.

For the concrete beam, some basic section properties like cross sectional area and moment of inertia were not accessible by the user, meaning that these properties were hidden from the user. This was not an issue in *RSA*, because after the transfer it was mapped to an appropriate profile in *RSA* and these properties were present in the *RSA* model. Furthermore, there was an option to send the reinforcement of structural beams and columns from *Revit* to *RSA*, however, even after multiple attempts and consulting the official Autodesk forums, they did not show up in *RSA*. On the other hand, the reinforcement from *RSA* could be transferred to *Revit*.

For the timber beam a similar issue was present as with the concrete beam, namely that for user defined sections, the cross sectional area and moment of inertia were not accessible. The 5% quantile of the modulus of elasticity was not included as a material parameter in *Revit*, and it was also not accessible in *RSA*, only after the timber design had been performed and could only be seen in the detailed calculation notes. Some minor differences also occurred in the values for other material properties like tensile strength perpendicular to grain, however these were so small that they can be considered negligible. Finally, the tree species from *Revit* could not be transferred to *RSA*.

Regarding structural analysis and design, there were a few additional modifications that needed to be made in the FEM software. Naturally, the first step was to make sure that the transferred information was correct and the model could be used as planned. After this, the necessary modifications could be done based on the intended design method.

For the steel beam no modifications were needed to run the structural analysis. Adjustments could be made about the number of finite elements to divide the beam into, but it was also done automatically by *RSA*. To perform a code check based on Eurocode 3, several design parameters needed to be set in *RSA* such as buckling length, load type, length coefficient for LTB, etc. However, these kind of parameters are only needed by the structural engineer so there would be no need to export/import these.

The same applies for the concrete and timber beam, with the addition for the concrete beam that the parameters for the reinforcement also need to be input to be able to run design checks.

5.1.3 StruXML

Model transfer from Tekla Structures Learning to FEM-Design and encountered issues

The geometry of the analytical model, mapped cross-section of the beam, the mapped material properties of the beam and the grid system was transferred, in addition to the end and start releases which are used to specify internal release conditions relating to the ends of structural members (fixed, pinned, user defined). Loads and supports were not transferred, because these were not supported with this link. Moreover, it was not possible to transfer the position of the analytical line.

Generally, it is not easy to see what the cross-sectional parameters e.g. height, width, web thickness, flange thickness etc. are in *FEM-Design*. This information is only available after a design calculation is made. So in order to check which information is transferred from *Tekla Structures Learning* it is necessary to define loads and boundary conditions and do a design calculation.

For the steel beam elastic and plastic section modulus was not accessible in *Tekla Structures Learning*, and the same was the case in *FEM-Design*. Only the modulus of elasticity was visible to the user. Supports and loads were not possible to transfer which was a limitation of *Tekla StruXML Export* 1.1.003. These were added to the model in *FEM-Design* in order to do design calculation.

For the concrete beam the reinforcement was not transferred, which was designed in *Tekla Structures Learning* to see if it is possible to transfer this to *FEM-Design*. This was instead designed in *FEM-Design* after adding loads and boundary conditions.

There were some limitations considering the timber beam. There is no predefined timber material in the library of *Tekla Structures Learning* when using the environment for Denmark (there are predefined timber materials when using e.g. environment for US). When the U.S. environment was chosen it was possible to choose different timber types, but very limited material properties were assigned to it - this is inadequate to design a structure. This may be a limitation because a student version of *Tekla Structures* called *Tekla Structures Learning* was used in this project.

Once the StruXML file was imported in *FEM-Design*, the supports were added to the beam. The next step was to define the different load cases and make a load combination.

For the steel and concrete beam no further modifications were needed.

For the timber beam an additional step was necessary. In the mapping process the desired material was chosen, but it was not possible to choose the desired cross-section since this particular profile was not available in the *FEM-Design* library so a random pre-defined cross-section was chosen. After the model was imported in *FEM-Design*, the user had to create a new timber profile with the desired dimensions and modify the transferred cross-section.

Finally, the analysis was carried out, where it was possible to perform the calculation based on load cases or load combination. There is an option for design calculation, where the user can choose to check or auto design all structural elements based on a chosen standard. Although, the parameters for the specific design case have to be specified by the user.

Model transfer from Revit to FEM-Design and encountered issues

Profile, material of structural member, eccentricity, loads and boundary conditions (including end releases) were transferred using *StruSoft StruXML Revit Add-in*. It was noted that load combinations and self-weight were not transferred. Furthermore, certain section properties were also not transferred e.g. shear area.

After transferring the model from *Revit* to *FEM-Design* a design calculation was made in order to access section properties and material properties for all three types of materials.

For the steel beam no serious issues occurred during the transferring process.

Out of the three material types, the worst result was obtained with the reinforced concrete beam where no information about reinforcement was transferred. The consequence of this is that the user needs to model the reinforcement in the FEM software. Other than class of exposure and characteristic cube strength, almost all information about the concrete specified in *Revit* was transferred to *FEM-Design* through the *StruXML Revit Add-in* as shown in Table 5.6, Table 5.7 and Table 5.8.

For the timber beam the timber type and timber species were not transferred. Other than this no further issues were encountered.

In order to do the design calculation it is only necessary to add the self-weight and make a load combination. This procedure was performed for the steel, concrete and timber beam.

Compared to the StruXML connection between *Tekla Structures Learning* and *FEM-Design*, this connection is more developed and the user only needs to do minor modifications once the model is imported to the finite element software before the design calculation is carried out.

5.1.4 ISM

Model transfer from AECOsim to STAAD.Pro and encountered issues

The transfer method with ISM is a quite simple procedure. When the model is ready for the export, simply choosing the option to create the ISM repository is needed.

There were two main kinds of encountered issues during the use of Bentley products and the transfer between them, which are software and transfer issues.

Software issues

Regarding the software issues, one of the most important limitation to mention was the lack of the structural sections. For example, cross-sections, which were defined in this project in the beginning of this chapter, could not be chosen from the program's database. It is likely in real life, that in a specific software a desired section can not be found, but in such cases the user can define it. In *AECOsim* there is a possibility to create user defined sections, however, the procedure is really complicated and requires advanced coding knowledge. Due to these reasons user defined cross-sections could not be used in these case studies.

Another limitation from an engineering point of view, is that material grades, supports and loads also can not be defined in the program. From this, it can be concluded that *AECOsim* is a powerful drawing tool for architectural purposes.

Transfer issues

Concerning the transfer issues, as it can be seen on the following figure, the section that was defined in *AECOsim* was not available in *STAAD.Pro*.

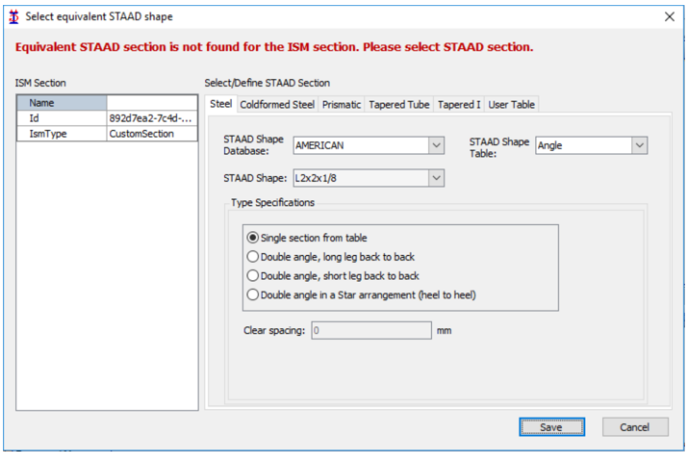


Figure 5.5: Unrecognised section during the import.

As it can be observed in Figure 5.5, *STAAD.Pro* offered a correction to select the desired beam section which couldn't be recognized before. When the correction for the unrecognised section was done, and the software finished the import, the result was not satisfying, because instead of an IPE beam section the outcome was a rectangular beam. This can be observed in Figure 5.6.

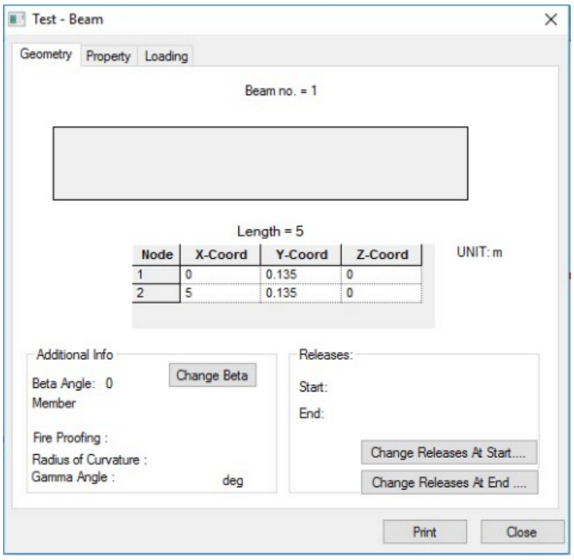


Figure 5.6: The result of IPE beam import.

The physical conditions of the concrete and timber beams, such as geometry, moment of inertia, cross-sectional area were imported correctly.

Loads and boundary conditions could not be defined in *AECOsim* and the import of these is not possible. Furthermore, because of the lack of the definition of material grades in *AECOsim*, the material properties needed for a structural analysis were set to zero. These values are indispensable to run a structural analysis, therefore manual correction is needed to run it.

In order the run a structural analysis in *STAAD.Pro* after importing a model from *AECOsim*, the properties for the entire model, except the geometry, have to be redefined. This includes the definition of the supports, creation of the load cases and the load combinations and the modification the section properties. After all these changes are made *STAAD.Pro* is able to make a structural analysis.

Additionally, Eurocode 2 is not included in the software, therefore the validation for the calculations can not be made. Design checks for steel and timber materials could be carried out after specifying their respective design parameters.

5.1.5 IFC

Model transfer from Tekla Structures Learning to RSA and encountered issues

The data transfer capabilities of IFC are quite robust, but great care should be taken when exporting the model to IFC so the data is transferred in the proper form.

Regarding the steel beam there was one fundamental issue that prevents an adequate examination. The problem was that *RSA* was not able to properly load the model from IFC and only showed an empty model. The IFC file was checked with *Solibri Model Checker* where it showed every information in the correct data structure. Additionally, the same IFC file was opened with *FEM-Design* where it was imported correctly. Based on these, the problem most likely lies with *RSA*, however, after many attempts to solving the problem and consulting the official Autodesk forums, the issue could not be resolved.

For the concrete beam the material and the section profile was transferred correctly to *RSA* but the element type was changed to bar instead of beam. This issue could be resolved by changing the element type manually in *RSA* after the import, which was a rather counter-productive solution as it was going against the automation process.

The timber beam model had the same issue with the structural type change as the concrete one. However, it also had an additional problem with the material properties. The timber material was not recognized in *RSA*, even though it was shown to be correct in *Solibri Model Checker*. Instead it was imported as concrete C25. It was thought that it could be caused by the different naming conventions of the various standards, so for this reason several material standards were examined both in *Tekla Structures Learning* and *RSA*, however this did not solve the issue. Thus, the material had to be manually changed after import.

With regards to structural analysis several different adjustments needed to be made for the IFC files to be able to properly run a structural analysis. Common to every model, the structural type needed to be changed from a bar to a beam element. Furthermore, loads, load combinations and boundary conditions were also specified manually in *RSA*. To perform code checks the design parameters also needed to be set for each model individually, just as it was described in Section 5.1.2 (p. 35).

As mentioned earlier the steel model could not be imported properly in *RSA* so obviously no structural analysis was done with that model.

For the concrete model the only modifications were the ones that were described above for every material.

Aside from the common changes the material also needed to be manually given for the timber beam.

Model transfer from AECOsim to RSA and encountered issues

There were numerous information that was not transferred to *RSA* preventing an accurate structural analysis.

Regarding the steel model, the structural type changed to bar in *RSA* from beam. An even bigger problem was, that the profile was not recognized by *RSA* during the transfer thus there were no profile assigned to the beam. Furthermore, the material of the beam was also changed from steel to C25 concrete.

For the concrete beam the only problem was the reoccurring one changing the structural type to bar instead of beam. The profile and the material was transferred correctly.

The timber model again experienced the issue where the structural type was changed to a bar. The material also changed to C25 concrete, while the profile was transferred correctly.

The IFC files for all materials were checked with *Solibri Model Checker* to try to identify the source of these problems. It showed that the issues were caused by the incorrect export of the IFC files from *AECOsim*. Many information were not transferred to the IFC file, like profile name for the IPE 270 beam, and others were exported but in an incorrect data structure, like the material for timber. This was probably caused by an incorrect mapping scheme for *AECOsim*, however the manual modification of this was not accessible to the project group, thus these issues could not be resolved.

For structural analysis, the same common changes had to be made for every model as what was described for the *Tekla Structures Learning - RSA* link, since most of them came from the limitations of the employed IFC scheme.

The difference compared to the *Tekla Structures Learning - RSA* case came only for the steel model, where now a section profile and a material needed to be assigned to the beam.

Model transfer from Revit to STAAD.Pro and encountered issues

For the model transfer first the IFC file had to be converted into ISM and then it could be imported into *STAAD.Pro*.

For all materials the same problems occurred as for the ISM link described in Section 5.1.4 (p. 38).

An additional issue arose when importing timber material into *STAAD.Pro*. The material grade and therefore all the properties were recognised as steel. Although, in the section database of the software, timber material could be found but only American and Canadian standards. It was assumed, that this issue was related to the lack of European timber elements.

The modifications needed to perform a structural analysis were the same for the IFC files as for the ISM case for the *AECOsim - STAAD.Pro* link in Section 5.1.4 (p. 38). For this reason they will not be described here again.

Model transfer from Tekla Structures Learning to STAAD.Pro and encountered issues

Model transfer, encountered issues and modifications for structural analysis were exactly the same like the *Revit - STAAD.Pro* data transfer, therefore detailed description for this part can be seen in the previous paragraph.

5.2 Simple Frame

To further investigate the capabilities of the data exchange methods a slightly more complex structural system was examined. This simple frame introduced columns and connections to be dealt with during the data transfer.

The analysed frame was a part of a frame system which can be seen in Figure 5.7. The single frame can be seen in red, as part of the complete frame system.

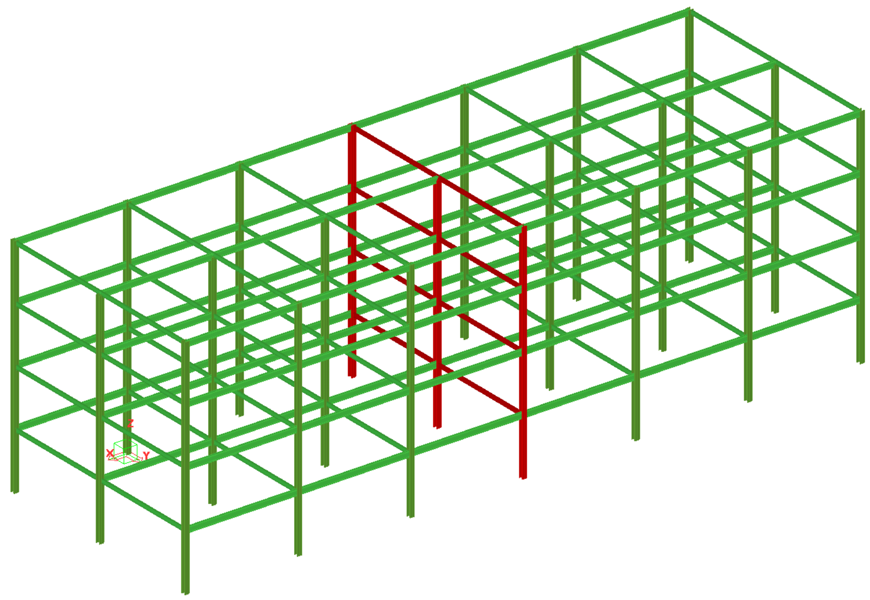


Figure 5.7: Complete frame system.

The physical model of the steel frame structure can be seen in Figure 5.8.

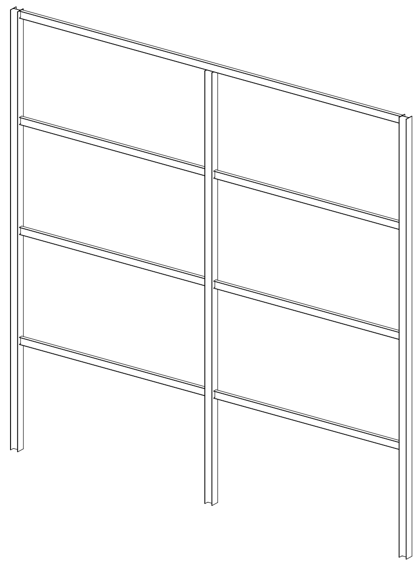


Figure 5.8: Physical model of the steel frame in *Revit*.

The used cross sections can be seen in Table 5.9. Material for steel and concrete was S 235 and C 25 respectively. For the frame system the reinforcement was not included in the models as it was found previously with the beam that the transfer of it did not work correctly.

Table 5.9: Used cross sections.

	Steel	Concrete
Beam	IPE 200	300mm x 450mm
Column	HEA 240	300mm x 400mm

The timber material was omitted as a frame structure like this would not be built from timber.

In Figure 5.9 a sketch of the the analytical frame structure is illustrated with dimensions and boundary conditions.

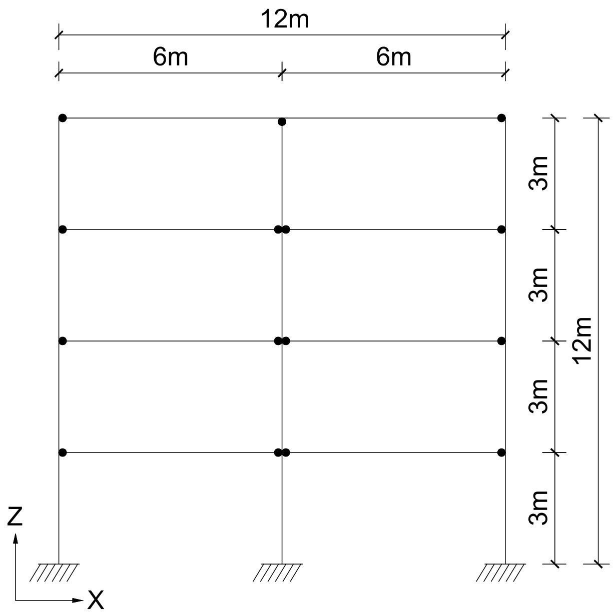


Figure 5.9: Analytical model of the frame.

By default the analytical lines are located at the center of structural members cross-section, but the user can change this as desired. In this case, the analytical model of the beam in the top is placed in the bottom center of its cross-section. Another solution could be to place the analytical model in the center of the beams cross-section and then simply move the beam down in order for the beams and columns analytical model to be connected. But this would be an improper solution because the CAD model then would not represent the actual geometry.

In the base the columns are fixed, so both translational and rotational degrees of freedom are fixed. At the top only translational degrees of freedom are fixed, so no moment can be transferred. The beam releases are defined identically in both ends with hinges, where only translational degrees of freedom are fixed. The load values are presented in Table 5.10 and the load positions are shown in Figure 5.10.

Table 5.10: Applied loads.

Load	Value
Wind load - w	7.8 kN/m
Live load - q	15 kN/m
Snow load - s	6 kN/m

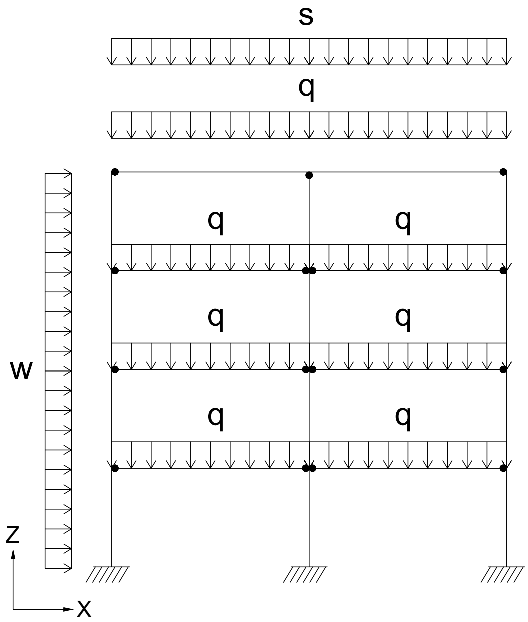


Figure 5.10: Loading of the frame.

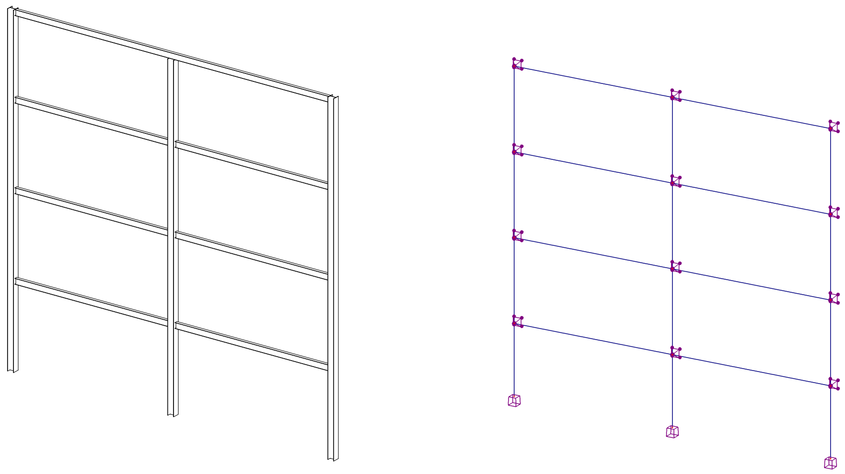
As for the simple beam, manual load combinations for the frame structure were also defined. The frame was treated as a middle frame from a frame system, thus there were point supports at every node, which prohibit movement of that node out of the plane.

In order to evaluate the link between CAD and FEM software, it was investigated if the user could perform analysis and stability check right after the import. Moreover, it was investigated what the user needed to change or add in order to prepare the imported model. The calculation made in FEM software included cross-sectional forces, displacement and stability check.

5.2.1 Revit to RSA direct link

Model transfer from Revit to RSA and encountered issues

The transfer process using the direct link between *Revit* and *RSA* was relatively simple compared to the data exchange using IFC or StruXML format. This exchange method was described in Section 4.1 (p. 19). The process of transferring the steel frame structure to *RSA* is illustrated in Figure 5.11. The procedure was naturally the same for the concrete structure.



(a) Physical model in *Revit*. (b) Analytical model in *RSA*.

Figure 5.11: Model transfer from *Revit* to *RSA* using direct link.

For the concrete frame structure similar issues were encountered as for the simple concrete beam. Geometry, cross-section of the beams and columns, material properties, boundary conditions, loads and the position of analytical line were transferred as specified in *Revit*. The only issue with the concrete frame structure was that certain material properties e.g. the cube strength were not accessible in *Revit* or *RSA*. Another important parameter was the tensile strength, which was defined in *Revit* but when transferred to *RSA* it was not accessible. Overall, the direct link between *Revit* and *RSA* was very accurate and capable relative to other options available to exchange data between *Revit* and *RSA*.

Regarding structural analysis, the user only needs to add the self-weight and then make a load combination containing the self-weight before the structural analysis can be performed for both models. Code checks for both materials can be performed after the proper design parameters have been specified.

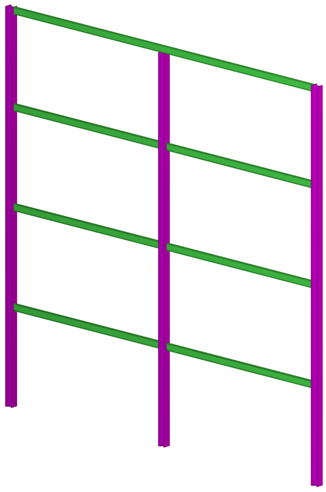
5.2.2 StruXML

Model transfer from Tekla Structures Learning to FEM-Design and encountered issues

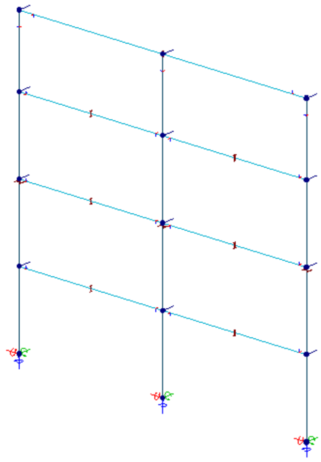
Aside from the loads and supports every information was transferred that is needed for the structural analysis.

There were no issues encountered during the data transfer. It worked as intended.

The model transfer can be seen in Figure 5.12



(a) Physical model in *Tekla Structures Learning*.



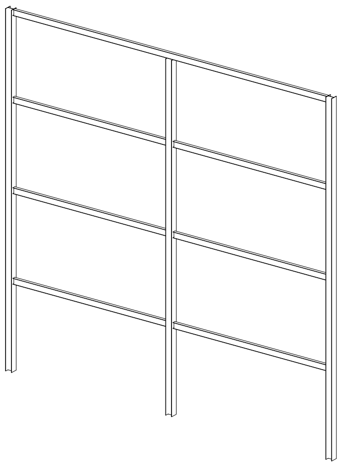
(b) Analytical model in *FEM-Design*.

Figure 5.12: Model transfer from *Tekla Structures Learning* to *FEM-Design* using StruXML.

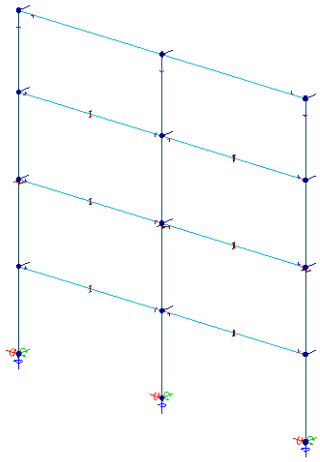
To be able to perform a structural analysis loads, load combinations and boundary conditions needed to be specified for both models. Additionally, the appropriate design parameters were also set for code checks.

Model transfer from Revit to FEM-Design and encountered issues

Every information, including the loads and supports, were transferred from *Revit* to *FEM-Design*. The model transfer is shown in Figure 5.13



(a) Physical model in *Revit*.



(b) Analytical model in *FEM-Design*.

Figure 5.13: Model transfer from *Revit* to *FEM-Design* using StruXML.

Since loads and boundary conditions were also transferred to *FEM-Design*, only the load combinations were needed to be defined to be able to perform a structural analysis.

5.2.3 IFC

Model transfer from Tekla Structures Learning to FEM-Design and encountered issues

Aside from the boundary conditions and loads, every other information was transferred to FEM-Design to be able to perform a structural analysis.

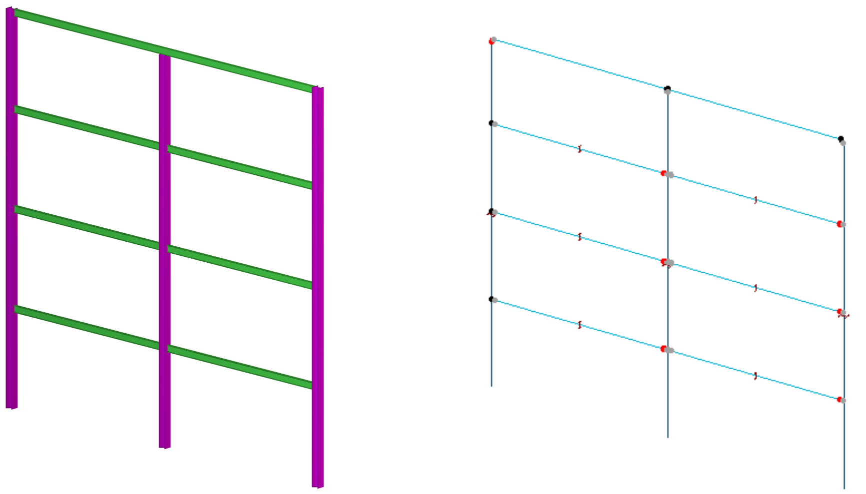
For the steel model the biggest issues were with the connections between the columns and the beams. In the imported model the nodes were replaced by a gap and rigid link to connect the elements. This was caused by the actual geometry of the structure where the end of the beams were cut back resulting in their analytical model not connecting directly to the analytical model of the columns. The analytical model in *Tekla Structures Learning* was crated correctly, however due to the limitations of the IFC2x3 Coordination View 2.0, the analytical model itself could not be exported, only the physical model. As a not ideal solution the *Tekla Structures Learning* model can be modified so that the cut backs are removed. This results in a correct analytical model in *FEM-Design*, but raises a problem in *Tekla Structures Learning* where it no longer represents the actual structure.

The profile of both the beams and columns were transferred correctly with the proper properties, but *FEM-Design* did not recognize them as an equivalent of a section in its own library. This meant that the profiles were there but they were not listed as IPE 200 for example, instead they were shown as a separate undefined section. This problem did not prevent structural analysis as the properties of the undefined sections were correct.

There were no issues with the material of the elements, and there was also an additional mapping option at the import process in *FEM-Design* to make sure that the material is imported correctly.

The concrete model experienced the same problems with the column beam connections as the steel model. However, the concrete profiles were identified correctly and were shown under the proper *FEM-Design* library section. The concrete material was also properly transferred.

The model transfer can be seen in Figure 5.14



(a) Physical model in *Tekla Structures Learning*.

(b) Analytical model in *FEM-Design*.

Figure 5.14: Model transfer from *Tekla Structures Learning* to *FEM-Design* using IFC.

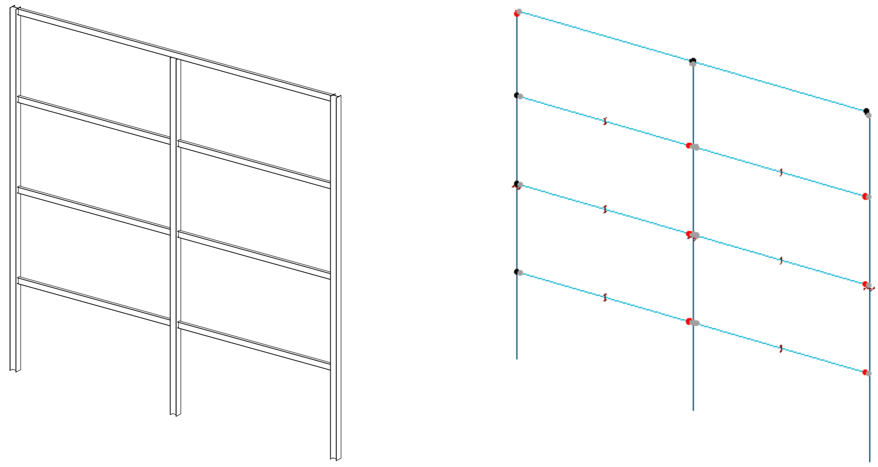
Due to the incorrectly imported analytical model, there are several modifications that needed to be made. Firstly, the connections between the columns and the beams were corrected and a continuous analytical model was created. The loads, load combinations and boundary conditions were also specified, in addition to the design parameters for code checks.

Model transfer from Revit to FEM-Design and encountered issues

Regarding the data exchange the same information was transferred as with the IFC file created by *Tekla Structures Learning*.

For the *Revit* IFC files every issue was the exact same and occurred in the same form as for the *Tekla Structures Learning* IFC files, thus these are not described again. Details can be seen in the previous section.

The model transfer is shown in Figure 5.15



(a) Physical model in *Tekla Structures Learning*.

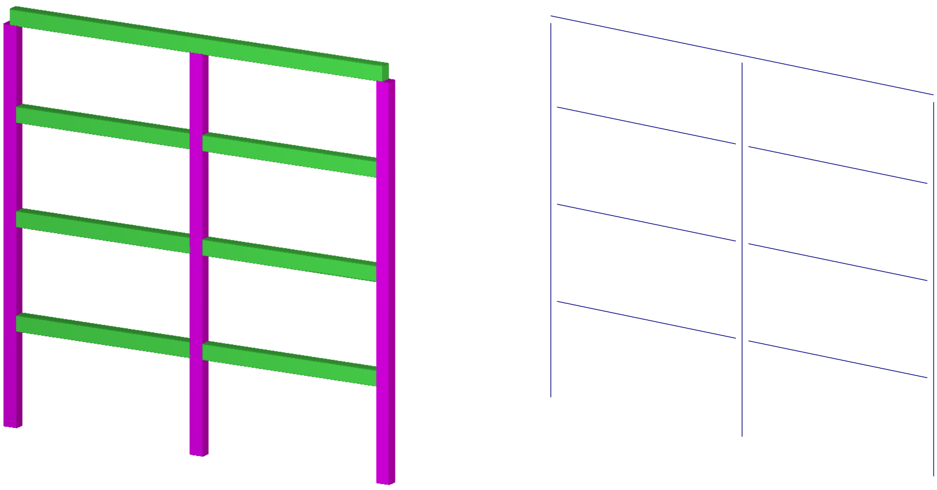
(b) Analytical model in *FEM-Design*.

Figure 5.15: Model transfer from *Revit* to *FEM-Design* using IFC.

The required modifications to perform a structural analysis are the same as with the *Tekla Structures Learning* - *FEM-Design* IFC link. Thus it will not be described here again, please refer back to the previous section for details.

Model transfer from Tekla Structures Learning to RSA and encountered issues

The frame structure was also exported from *Tekla Structures Learning* to *RSA* using the IFC format. The process is illustrated in Figure 5.16 for the concrete frame structure. The process is identical for the steel frame structure.



(a) Physical model in *Tekla Structures Learning*.

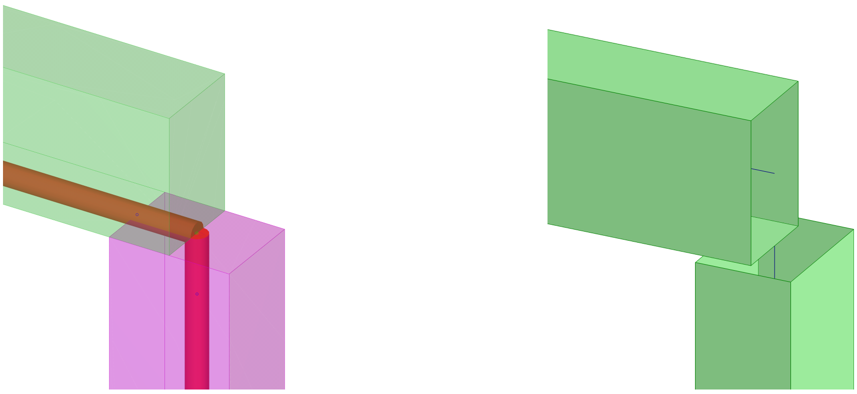
(b) Analytical model in *RSA*.

Figure 5.16: Model transfer from *Tekla Structures Learning* to *RSA* using IFC.

After transferring the steel frame structure from *Tekla Structures Learning* to *RSA*, the issue was that the model was empty. This was the same issue that occurred when dealing with a simple steel beam. The reason for this is not known.

For the concrete frame structure, as it is seen in Figure 5.16 that there were some issues with the connection between the structural elements as they were not connected. When exporting the model from *Tekla Structures Learning* the IFC2x3 Coordination View 2.0 was chosen, where it was not possible to send the structural analysis model which contains loads, connections and boundary conditions.

There was no connection between the analytical models in what was imported into *RSA*, which can be seen in Figure 5.17.



(a) Analytical model in *Tekla Structures Learning*.

(b) Analytical model in *RSA*.

Figure 5.17: Model transfer from *Tekla Structures Learning* to *RSA* using IFC.

The analytical model was not transferred, only the physical model. Due to the cut backs in the physical model, the different structural elements were not connected. Before the user can perform a structural analysis the model needs to be prepared by connecting the structural elements. This was done by elongating the columns so they are connected to the upper beam, and the beams needed to be elongated to be connected to the columns.

After modifying the structural system, the next step is to define boundary conditions, loads and make load combinations. Finally, the structural analysis can be carried out.

5.3 Discussion of simple cases

This section will provide an evaluation of the case studies for the simple beam and frame structure for each data exchange method separately.

5.3.1 Revit to RSA direct link

Based on the data gathered from the case studies for the simple structures it can be stated that the *Revit - RSA* direct link is the best of the data exchange methods that were examined both in terms of technical capabilities and ease of use. It is able to transfer every information including loads and boundary conditions that are needed to perform a structural analysis.

The ease of use is also exceptional as the transfer is easily initiated and done in a data exchange window in *Revit*. *RSA* is launched automatically as part of the process and the model is ready there in basically a few seconds.

Furthermore, a very useful feature of it is that it is bidirectional, which makes it easy and convenient to implement changes in the CAD model based on the results of the structural analysis. The data transfer can just as well be started from *RSA* and the *Revit* model will be updated according to the *RSA* model.

The one drawback of this link is that it only works with these two specific applications. Moreover, to utilize its full potential both applications need to be installed on the same computer. However, considering that both *Revit* and *RSA* are the market leaders in their respective fields, it is reasonable to say that this direct link has a very high potential in practical applications.

5.3.2 StruXML

The StruXML format is developed by StruSoft and is used to transfer data from *Tekla Structures Learning* to *FEM-Design*, and to exchange data between *Revit* and *FEM-Design*. Considering the results from the case studies, the StruXML format is a useful and capable method to exchange data. Especially, for the link between *Revit* and *FEM-Design* where the transfer process using *StruXML Revit Add-in* is a better equipped link than the connection between *Tekla Structures Learning* and *FEM-Design* using *Tekla StruXML Export*, since also boundary conditions, loads and position of analytical lines can be exchanged. Bidirectional data transfer is another advantage of StruXML between *Revit* and *FEM-Design*.

The process of data exchange is not fully automated, since the user needs to map materials and cross-sections to the library of *FEM-Design*. This can cause issues, if the chosen material or cross-section in either *Tekla Structures Learning* or *Revit* does not exist in the library of *FEM-Design*.

Aside from the above mentioned drawbacks, StruXML is a very useful format which enables data exchange that includes information needed to perform design calculations. Compared to the IFC2x3 Coordination View 2.0 StruXML has the advantage of also transferring boundary conditions and loads. In case of model transfer between *Tekla Structures Learning* and *FEM-Design* the user needs to do some significant changes in the transferred model before design calculations can be made, whereas there is only need for few changes to be made for a model transferred from *Revit* to *FEM-Design* e.g. add self-weight and create load combinations.

5.3.3 ISM

Although, the ISM format was only used at the simple beam part, it can be stated that this link has the most limited capability to transfer data form a CAD software to a FEM application. This link can only transfer geometrical, cross sectional and material property data to a FEM software. In *AECOsim* detailed material properties such as grade and Young's modulus could not be defined, only a material name could be given for the parts. Therefore the interoperability of these two software were even more limited.

It is also noted during the use of the Bentley products, that the European code standards are not implemented well in the software. Meaning that, for example Eurocode 2 is missing from the database. However, it is still possible to perform a reinforced concrete design, but with another code.

Moreover, the section database contains a wide range of American standards while the European standards are limited. This can cause an issue with the data transfer because the software does not recognize the elements from another standard other than American, which was the issue at this project also.

5.3.4 IFC

The technical capabilities of the IFC data exchange method varies a lot based on the application it is created, the application it is opened with and the material used.

It is a common trait that the IFC file is created and saved as a separate file which can then be opened with the FEM software. However, the type of information and the way it is exported differs from software to software. A key aspect of this is the supported MVDs of the applications. Comparing *Revit*, *Tekla Structures Learning* and *AECOsim*, it is seen that *Revit* has the most MVDs implemented and it is the only software that supports IFC4 at this time. It is also straightforward to use custom property sets with it, and create customizable MVDs based on existing ones. *Tekla Structures Learning* also allows for easy usage of custom property sets but has fewer MVDs supported, which limits the capabilities of an IFC file created with it. However, this would only mean an actual problem in very rare and specific cases as the MVDs that are most widely used are accessible in *Tekla Structures Learning*. With regards to *AECOsim* it also has fewer MVDs supported than *Revit* and during the tests many errors are experienced with the IFC export. A lot of the data is incorrectly exported to the IFC files and the manual adjustment of the mapping is either inaccessible or a very confusing process.

The IFC exchange method has many issues with the proper data exchange. Ranging from difficulties with timber material, to *RSA* not recognizing correct IFC files and incorrect analytical models imported to FEM software. Some of these issues originate from incorrect export from the CAD software, while others from incorrect import to FEM applications. This reflects the unique challenges that are connected to IFC. As opposed to the *Revit* - *RSA* direct link, or the StruXML file, which are developed by Autodesk and StruSoft respectively for both ends of the data exchange link, the implementation of IFC standards are done by the respective software vendors separately. So in the data exchange process from *Tekla Structures Learning* to *FEM-Design* the IFC export and import are done by two separate companies without any coordination other than the certification standard for IFC.

Regarding the ease of use, IFC is a simple data exchange method provided that the default settings can be used. If custom property sets need to be included in the process it heavily hinders the ease of use as a user-defined mapping scheme needs to be created.

Another issue with IFC is the slow implementation of it. For one example the IFC2x3 Structural Analysis View was released eight years ago and it is still not supported by any of the software that was examined. For this reason, even though IFC has the technical capabilities to deal with a wide array of problems, it can not live up to its full potential.

However, the IFC standard and what it stands for is extremely important for the industry itself. As engineering projects get bigger and more complex open communication and data sharing between the participants is crucial for efficient and sustainable workflow. Not only does IFC allow for data exchange between different software vendors, but also between different disciplines. But to reach its full potential especially in the AEC sector it needs more support from software developers.

Part II

Advanced cases

Chapter 6

Data transfer methods

In this chapter the various data exchange links, shown in Figure 2.2 and used in the advanced cases part, are described briefly. A detailed description of the necessary steps is found in Appendix C.

These specific links were chosen, because this part was done in collaboration with two engineering companies, namely ISC Consulting Engineers and 3D Structural Design, who use these software. Thus the focus was on those links that are compatible with the applications the companies use.

6.1 Tekla Structures Learning - RFEM direct link

The process of data exchange can be seen in Figure 6.1

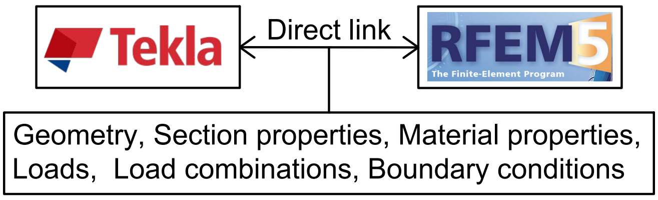


Figure 6.1: Data exchange between *Tekla Structures Learning* and *RFEM*.

The direct link between *Tekla Structures Learning* and *RFEM* is a useful method to exchange data and very easy to use. The data exchange process can be initiated either in *Tekla Structures Learning* or in *RFEM*. The requirement for this to work is that both software applications are installed on the same PC and running simultaneously.

It is possible to transfer both structural data and load data. The structural data includes members, member releases, nodal supports, eccentricities, rigid connection, etc., while the load data includes load cases and load combinations. Furthermore, it is possible to reimport the model from *RFEM* to *Tekla Structures Learning* and update e.g. cross-sections.

A more thorough technical description can be found at Dlubal Engineering Software [2010].

6.2 CIS

The data exchange process can be seen in Figure 6.2.

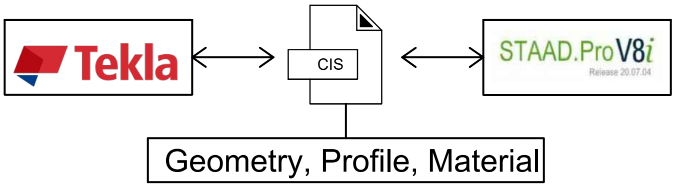


Figure 6.2: Data exchange between *Tekla Structures Learning* and *STAAD.Pro* using CIS format.

CIMSteel Integration Standards (CIS) is a data exchange file format for structural steel projects. In this project it is used to transfer data from *Tekla Structures Learning* to *STAAD.Pro*.

CIS format supports data exchange of members, material, member type, etc, but for this to work complete material and cross-section conversation files are required. The material conversation file is used to map the cross-sections in *Tekla Structures Learning* to cross-sections in *STAAD.Pro*. Similarly, the material conversation file is used to map material data.

More information about CIS is available in American Institute of Steel Construction [2016].

6.3 ISM

This section introduces the chosen link between *ProSteel*, *STAAD.Pro* and *RFEM*. The transfer format between the three software is ISM. Since, *ProSteel* and *STAAD.Pro* are Bentley products, they can communicate with ISM. Moreover, *RFEM* is also capable of exporting/importing ISM, so it has been decided to use this format during the investigations. A description about the basic features of ISM can be found in Section 4.3 (p. 21).

6.3.1 ProSteel - STAAD.Pro

The general process for the data exchange can be seen in Figure 6.3.

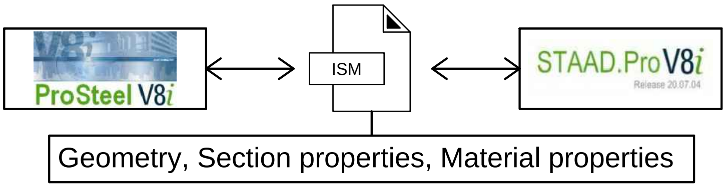


Figure 6.3: ISM link between *ProSteel* and *STAAD.Pro*.

6.3.2 ProSteel - RFEM

Similarly, the exchange method can be seen in Figure 6.4.

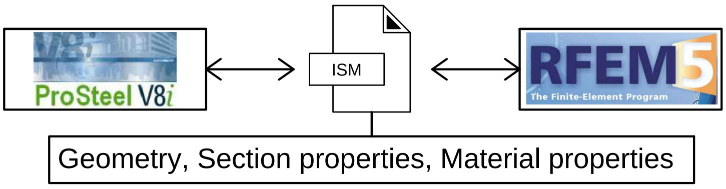


Figure 6.4: ISM link between *ProSteel* and *RFEM*.

The steps to perform the data transfer from *ProSteel* to *STAAD.Pro* and *RFEM* can be found in Appendix C.

6.4 Tekla Structures Learning - Abaqus/CAE

The process of data exchange between *Tekla Structures Learning* and *Abaqus/CAE* is illustrated in Figure 6.5. The necessary steps are described in Appendix C.7.

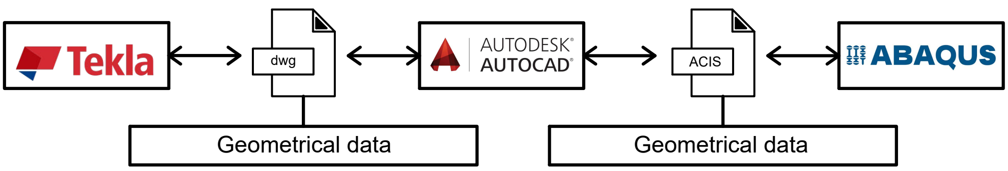


Figure 6.5: Data exchange between *Tekla Structures Learning* and *Abaqus/CAE*.

The file format dwg is a common file format for Autodesk products, in this case *AutoCAD*.

It was used to exchange geometrical data between *Tekla Structures Learning* and *AutoCAD*. The exchanged model was first transferred to *AutoCAD* as a Polyface Mesh, and in order to convert this to 3D solid, the model needed to be meshed geometrically. Then, using Convert to Solid feature, the model was converted to 3D solid. Furthermore, the 3D solid model was combined to just one part. This makes it easier to work with the model in *Abaqus/CAE*, because the user can avoid defining material and cross-section for each part, assemble the model and define constraints between the different parts.

In *AutoCAD* it is possible to export 3D solid to an ACIS file in ASCII (SAT) format. ACSII is an acronym for American Standard Code for Information Interchange, while SAT stands for Standard ACIS Text. This file format is robust and enables transfer of 3D models from CAD software, including wireframe, surface and solid models. [Spatial Corp., 2016] In this project this open format is used to transfer a 3D solid model from *AutoCAD* to *Abaqus/CAE*. Other objects, such as lines and arcs, are ignored. SAT files can also be opened with text editing software like *Notepad* and the user can read the content of the data file.

Chapter 7

Case studies - General analysis

This chapter describes the investigation of two building models with general purpose FEM software and analyses. The purpose of the investigation is to see how the inclusion of the CAD model in the design process provides actual benefits for it, if any. To investigate this, the CAD models of two structures, which were supplied by two engineering companies, ISC and 3D Structural Design, were used to create FE models. A structural analysis was performed on them and the results were compared to results from separately made FE models, also supplied by the aforementioned companies. The FE models made by these companies were built entirely in their respective FEM software thus by comparing the results with the ones that were done by the project group using the CAD data, the effect of incorporating it in the design process can be examined.

7.1 Platform structure from ISC

7.1.1 Model description

The following building model was given by ISC. The building is a steel construction and located in Norway. On Figure 7.1 the structure can be seen.

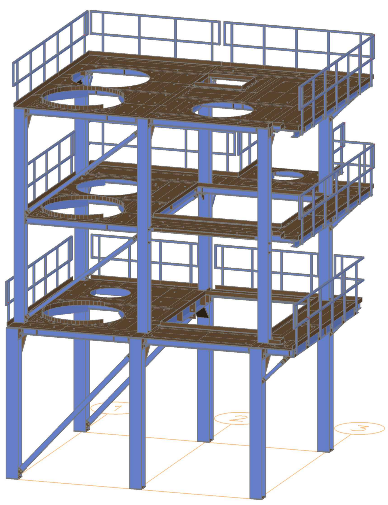


Figure 7.1: Platform structure given by ISC.

The building is approximately 5.5 m long, 7.2 m wide and 10 m high. As it was mentioned before, it is placed in Norway, inside an already existing building, therefore wind and snow loads are not acting on it. In Table 7.1 the used cross sections of the structure are listed.

Table 7.1: Used cross sections.

Columns	Beams	Bracing
HEA 300	HEA 160	RQ 120x4.5
HEA 260	IPE 220	RQ 80x3.6
	UNP 200	

Table 7.2 represents the material grade for the used cross sections.

Table 7.2: Material grade.

	Steel grades
Columns and beams	S355
Plates and handrails	S235

7.1.2 Transfer methods and encountered issues

ProSteel - STAAD.Pro

The exchange method is done in the same way as it was described in Section 4.3 (p. 21). When the import process is done through *Structural Synchronizer* with ISM, the analytical model is obtained, which can be seen on Figure 7.2.

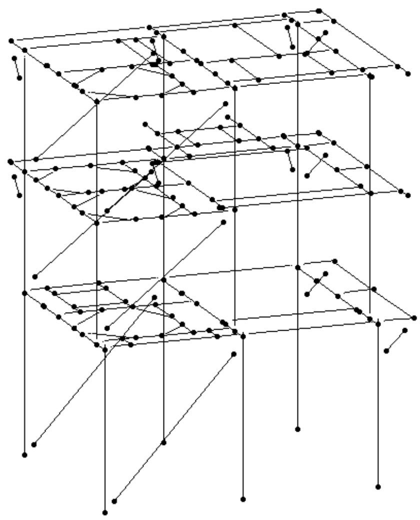
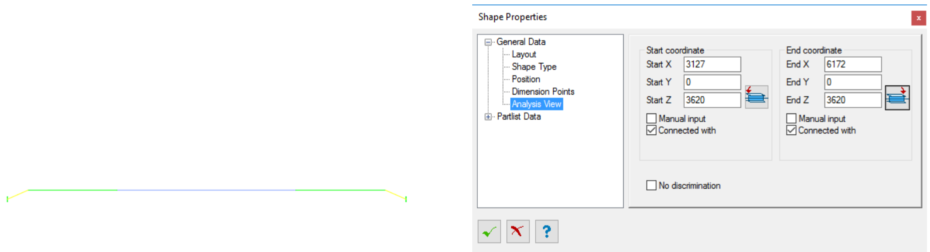


Figure 7.2: The analytical model of the structure in *STAAD.Pro*.

As it can be seen in the figure above the analytical model has gaps at the connections and at the bracing. These gaps are occurring when there are detailed connections in the CAD model or the connecting beams centreline are not at the same height. In these cases *STAAD.Pro* substitutes the gaps with rigid links. Rigid links are imaginary, weightless links between elements that can couple degrees of freedom of the connected members. They are used when the offset between the elements need to modeled and have a significant effect on the structural analysis. This is a common method among CAD software to handle gaps between elements, however in most cases a direct connection between the elements is the proper way to model the structure. Thus there were several attempts to fix the issue with the rigid links for this model.

The first one was in *ProSteel*, where the user has an option to highlight the analytical view for the model. In the view, the connection coordinates, the length and the height of the analytical lines were modified. Unfortunately, in this view only one beam can be highlighted at the time. To connect all the analytical lines the right coordinates were set up for each element separately. Figure 7.3 shows the analytical view option.



(a) The analytical view. (b) The coordinates.

Figure 7.3: The analytical model of one beam in *ProSteel* and its coordinates.

Unfortunately, this method did not solve the problem with the rigid links and gaps.

The second attempt was in *STAAD.Pro*. The program automatically generates offsets for the elements. These offsets were deleted after the import process. However, the bracing was still not connected, and there are some remaining nodes which were also not connected, and had to be connected manually.

Furthermore, the sections and the material properties of the bracing were not recognised during the import, which means these sections were transferred as a rectangular beam with the material properties set to zero, instead of a square hollow section with the steel grade of S235. The other sections were recognized correctly but the material properties were not. All the beams and columns were set to steel grade S355 in the CAD software. In *STAAD.Pro* only the name of the grade matches. The material properties were set to zero. These values also had to be defined manually. These issues are thought to be caused by either the differences in the libraries and different naming schemes of the two applications and/or the inability of *Structural Synchronizer* to properly map them. Moreover, as it was already experienced in Section 5.1.4 (p. 38) the supports are not transferred with ISM to the analytical model. This also has to be defined by the user in *STAAD.Pro*.

When all the modifications were done, the structural analysis was executed. Figure 7.4 shows the modified analytical model.

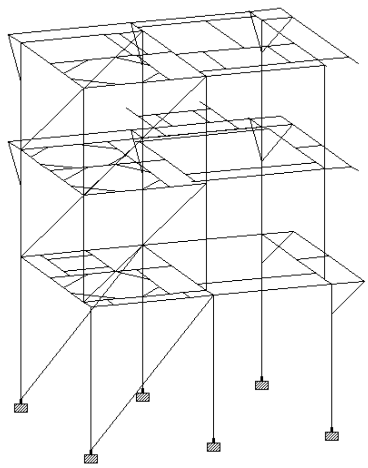


Figure 7.4: Analytical model after all the corrections in *STAAD.Pro*.

ProSteel - RFEM

The import process produced a correctly set-up analytical model. *RFEM* has an option to ignore the rigid links during the import method, which means, that after the import all the sections are connected to each other properly. This can be seen in the figure below.

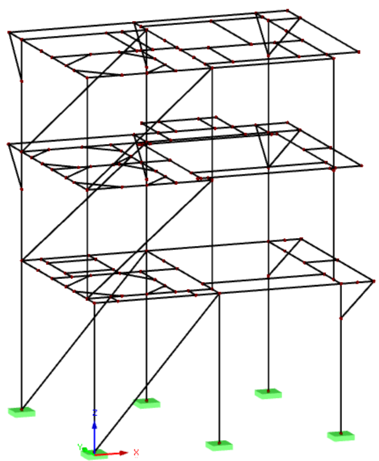
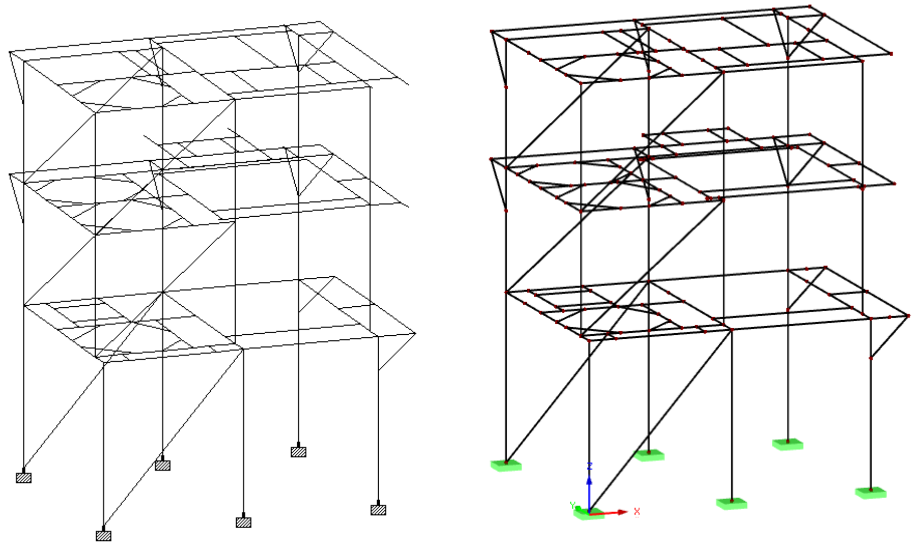


Figure 7.5: Analytical model after import in *RFEM*.

The supports in Figure 7.5 are added to the model after the import process. All beam and column parts are recognized during the import with the corresponding material and geometrical properties. Although, there was one issue with the bracing. The sections for those members were not recognized during the import, but this issue could be easily fixed afterwards. With the corrected sections the analytical model is prepared for a structural analysis.

7.1.3 Structural analysis

Both the *STAAD.Pro* and *RFEM* models contain the same supports, load distributions and load combinations as ISC provided in their own report. The load values, placement and combinations can be found in the Appendix DVD *Advanced cases/Platform structure from ISC/Design report1.pdf* and *Design report2.pdf*. The structure is modeled with beam elements only and every beam element is connected to each other with a moment resisting connection with all six degrees of freedom fixed. The supports are also fixed. The two FE models can be seen in Figure 7.6.



(a) Analytical model in *STAAD.Pro*.　　(b) Analytical model in *RFEM*.

Figure 7.6: Different analytical models in FEM software.

After the two FE models were set up correctly, a first order structural analysis was performed, using a linear elastic material model for steel. The results are compared to the results given by ISC. The discussion about the results can be found in the next section.

7.1.4 Results and comparison

In this section the results of the two FE analyses are presented. The compared results are the displacements and the support reactions. The results provided by ISC were calculated in a FEM software called *GT STRUDL*.

STAAD.Pro results

The obtained results from *STAAD.Pro* were completely incorrect due to the small gaps between the elements. This rendered the model unusable for any further analysis or comparison. The displacements of the structure can be seen on the figure below.

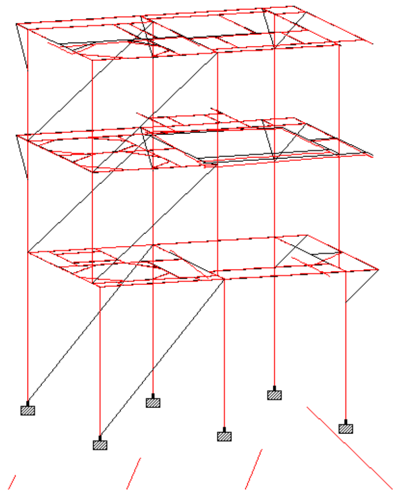


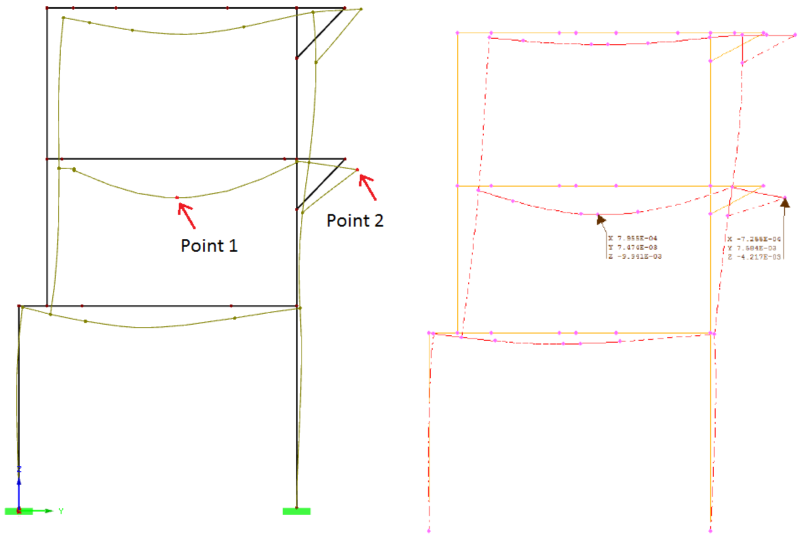
Figure 7.7: Result of displacement in *STAAD.Pro*.

From Figure 7.7 it can be seen that the system treated the bracing separately from the entire model. Although, the bracing seemed to be connected to system with nodes. Moreover, attempts were made to change the end releases of the bracing from rigid to pinned, where the rotational degree of freedom was fixed, but it could not solve this issue. Therefore, it was concluded that the results of *STAAD.Pro* were not acceptable.

RFEM results

During the analysis there were no issues with the bracing as it was in the previous section. The analysis was performed without problems, therefore the results could be compared with the *GT STRUDL* results. As it was mentioned earlier, the loads and load combinations can be found in the Appendix DVD *Advanced cases/Platform structure from ISC/Design report1.pdf* and *Design report2.pdf*.

For the displacement, ISC highlighted two reference points in their results. These two points were used to check the displacement in the *RFEM* model, shown in Figure 7.8.



(a) Displacement in *RFEM*. (b) Displacement in *GT STRUDL*.

Figure 7.8: Calculated and given displacement in comparison.

The results from the *RFEM* model and their comparison with the ISC result can be seen in Table 7.3.

Table 7.3: Displacement results [mm].

Point number	*RFEM*	*GT STRUDL*
1	8.4	9.94
2	3.5	4.22

Regarding the support reactions, Table 7.4 contains the result from *RFEM* and *GT STRUDL*. For a better overview the numbers of the supports can be seen in Figure 7.9.

Table 7.4: Support reactions. [kN]

Point	*RFEM*			*GT STRUDL*		
number	X	Y	Z	X	Y	Z
1	-2.23	6.45	86.43	12.8	9.8	86.4
2	-0.8	18.46	166.65	0.1	17.4	171.2
3	1.18	14.69	133.15	1.7	20	172.5
4	0.43	-14.67	223.48	1.7	5.3	279.8
5	-1.7	-16.2	290.50	0.1	2.2	331.5
6	3.18	-9.37	191.13	28	2.8	197.4

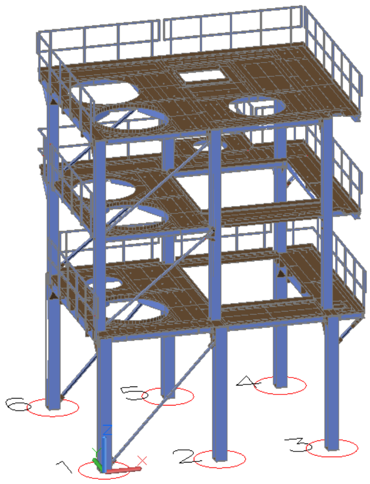


Figure 7.9: Numbers of the supports in *ProSteel*.

It can be seen from the results, that the deformation shape for the *RFEM* model follows the shape of the *GT STRUDL* model fairly well. There are bigger differences in the support reactions, however. This is due to the fact that the analytical model created by ISC was different than the analytical model created from *RFEM*. Namely, the diagonal bracing on the second and third floor connected the opposite corners, and the beam allocation was different in the *GT STRUDL* model than in the *RFEM* model.

7.2 Bridge structure from 3D Structural Design

7.2.1 Model description

A 3D CAD model of a conveyor bridge created in *Tekla Structures* and a 3D FE model of the conveyor bridge created in *STAAD.Pro* were provided by 3D Structural Design. The 3D CAD model is shown in Figure 7.10.

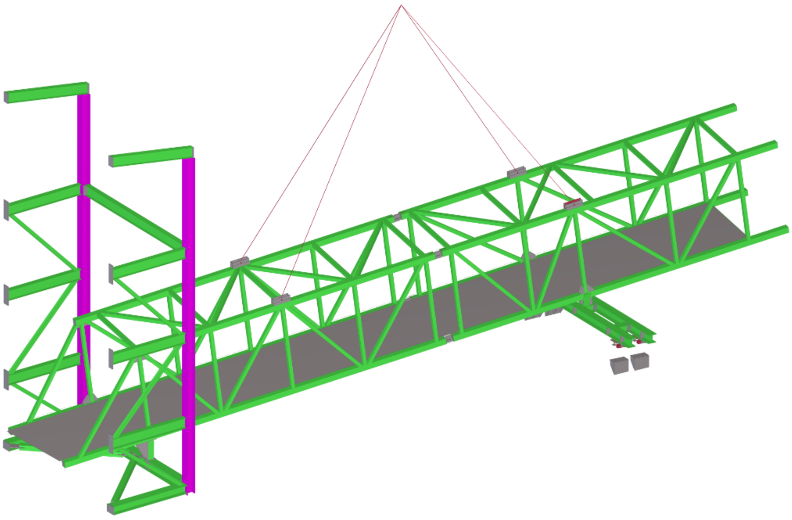


Figure 7.10: Physical model of the conveyor bridge.

The 3D model includes detailed information about connections, which is shown in Figure 7.11 and Figure 7.12.

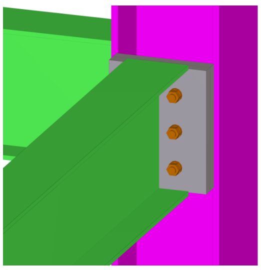


Figure 7.11: Bolted connection.

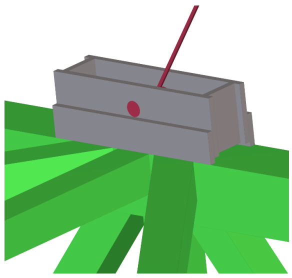


Figure 7.12: Welded connection.

The structure is made out of steel beam and column parts with several different cross-sections and a steel grade of S253. In Table 7.5 the different cross-sections in the model can be seen. This information is available to the project group through the 3D CAD model provided by 3D Structural Design.

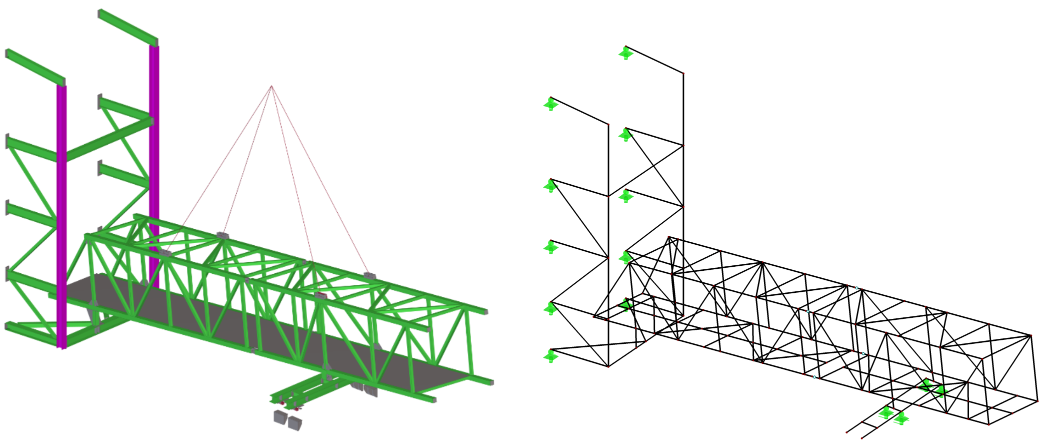
Table 7.5: Used cross-sections.

Beams		Columns
120X8SHS	180X6.3SHS	HE300B
200X8SHS	HE300B	
IPE360	120X6.3SHS	
HE360A	HE260B	

7.2.2 Transfer methods and encountered issues

Tekla Structures Learning - RFEM

The analytical model was imported from *Tekla Structures Learning* using the direct link. The import process is shown in Figure 7.13. The supports shown in the figure were added in the FE model in *RFEM* after the model was imported.

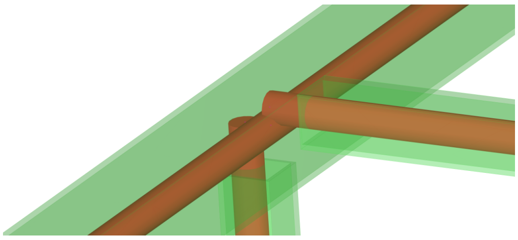


(a) Physical model in *Tekla Structures Learning*.

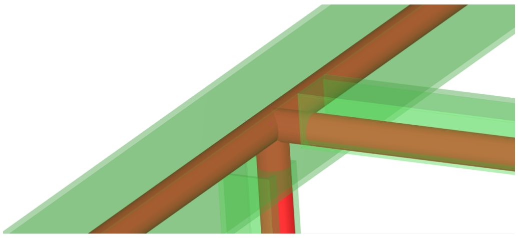
(b) Analytical model in *RFEM*.

Figure 7.13: Import from *Tekla Structures Learning* to *RFEM* using direct link.

A crucial task before transferring the model was to make sure that the Analysis Model was set-up as intended. The Analysis Model had been generated automatically based on the physical model first, and then the individual parts and nodes were modified. An example of the required modifications can be seen in Figure 7.14. It is apparent that on the left side the nodes are not aligned as they should be so they were merged to form just one node.



(a) Incorrect node.



(b) Correct node.

Figure 7.14: Incorrect and correct Analysis Model in *Tekla Structures Learning*.

Close attention had to be paid that the merging of the nodes to each other and the analysis line was done correctly, because an incorrect snap could result in a very small gap between the elements that is only visible with an extremely high zoom.

The data exchange between *Tekla Structures Learning* and *RFEM* is very well developed and very capable. Cross-section, member type, material and end releases were transferred. Supports, load cases and load combinations were added in the the project groups FE model, based on the FE model handed to the project group by 3D Structural Design. However, there were issues with two structural elements. They were defined as beam elements in *Tekla Structures Learning*, but were transferred as bar elements into *RFEM*. The two mentioned beams are shown in Figure 7.15 with red color.

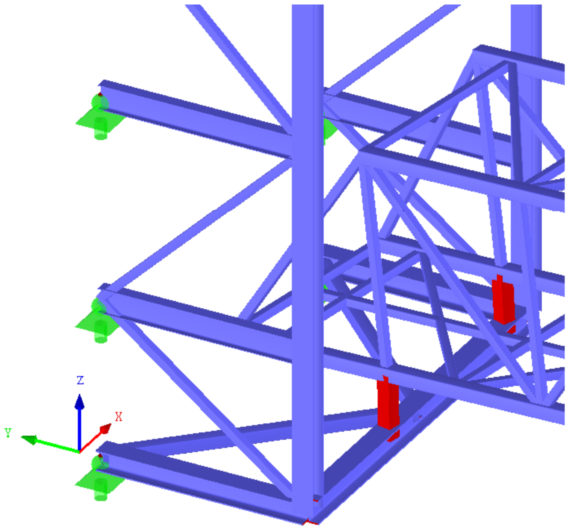


Figure 7.15: Two beam elements transferred as bar elements.

Once the self-weight was added and a calculation was run, the structure simply slided backwards in the y-direction. This issue was solved by changing the element type of the two structural elements from truss to beam.

Another issue was connected to the hinged connections in the bridge. In *Tekla Structures Learning* the two beams' end releases were defined with the rotational degrees of freedom about x-axis and y-axis released in local coordinate system c.f. Figure 7.16, based on the FE model from 3D Structural Design.

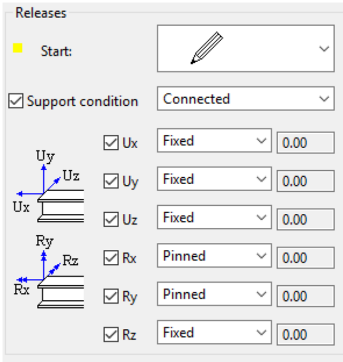
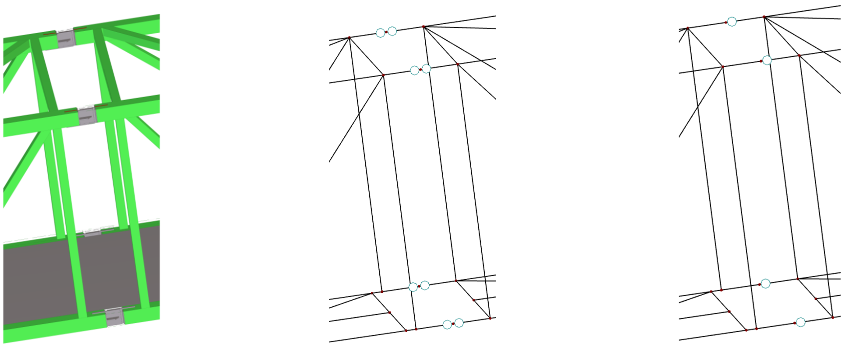


Figure 7.16: Definition of hinge in *Tekla Structures Learning*.

After the model was imported from *Tekla Structures Learning* to *RFEM*, supports and loads were added, and a calculation was run. An error occurred with the hinges in the middle of the beam, which were replaced with fixed end releases, as shown in Figure 7.17. This error was due to the way hinged connections are defined in *RFEM*.



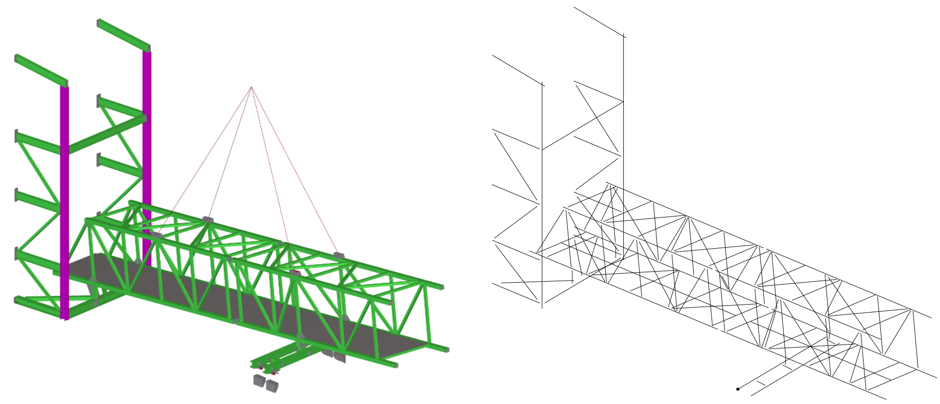
(a) Hinges in CAD model. (b) Transferred hinges. (c) Corrected hinges.

Figure 7.17: Data exchange from *Tekla Structures Learning* to *RFEM* using direct link.

Finally, a calculation was run with no issues. The results are presented and compared with the results from FE model from 3D Structural Design later in this chapter.

Tekla Structures Learning - STAAD.Pro

The 3D model of the conveyor bridge in *Tekla Structures Learning* was exported to the CIS/2 format. The import process is shown in Figure 7.18



(a) Physical model in *Tekla Structures Learning*.

(b) Analytical model in *STAAD.Pro*.

Figure 7.18: Data exchange from *Tekla Structures Learning* to *RFEM* using direct link.

Even though 'Analysis Model' was chosen during the export process from *Tekla Structures Learning*, once the model was imported in *STAAD.Pro*, the structural elements were not connected. There was a small gap between structural elements as shown in Figure 7.19.

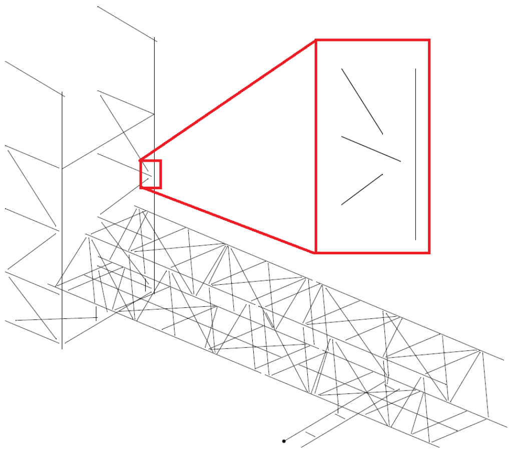


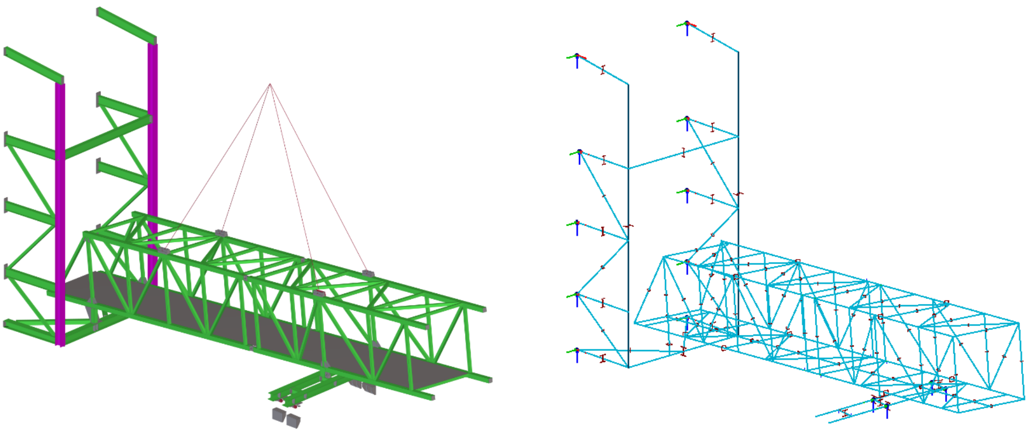
Figure 7.19: Gap between structural elements in *STAAD.Pro*.

Furthermore, another issue was that only IPE-sections were recognised in *STAAD.Pro* and material data was not transferred. Material type and cross-section type is defined differently in the library of *Tekla Structures Learning* and *STAAD.Pro*. This is why these properties were not recognized in *STAAD.Pro*.

Since it is time-consuming to connect the structural elements and assign structural elements profile and material in *STAAD.Pro*, which defeats the purpose of transferring the model from *Tekla Structures Learning*, it is decided not to work with the transferred model in *STAAD.Pro* further.

Tekla Structures Learning - FEM-Design

To transfer the CAD model of the bridge from *Tekla Structures Learning* to *FEM-Design* two methods were considered, namely StruXML and IFC. A significant advantage of StruXML over IFC is that it can easily transfer the Analysis Model from *Tekla Structures Learning*, which is essential in creating a correct FE model in *FEM-Design*. Having the Analysis Model properly transferred can greatly reduce the FE model creation time, thus StruXML was chosen for the data exchange. The model transfer can be seen in Figure 7.20. The supports were added in *FEM-Design* after the import.



(a) *Tekla Structures Learning* model. (b) *FEM-Design* model.

Figure 7.20: Model transfer using StruXML.

The Analysis Model in *Tekla Structures Learning* had to be adequately prepared to avoid gaps between the elements. Figure 7.21 illustrates these gaps that became apparent once the analysis was performed.

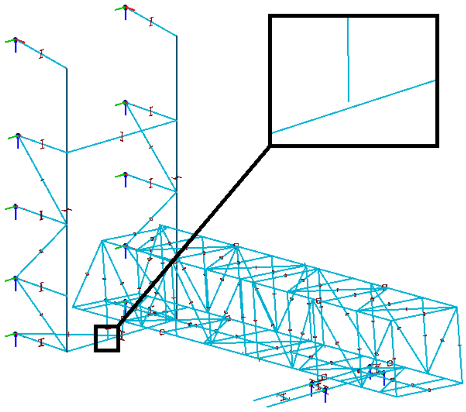


Figure 7.21: Small gap between elements in *FEM-Design*.

The transfer process was done as described in Section 4.2 (p. 20) with the help of the *Tekla StruXML Export* tool. There was one issue that occurred during the transport process, namely that one of the columns was exported incorrectly into the StruXML file. Due to this error the file could not be opened in *FEM-Design*. However, the file could be edited in a text editor and the incorrect column part could be deleted from it. After this it could be opened with *FEM-design*, with the column obviously missing from the model. This meant that particular column needed to be modeled again in *FEM-Design*.

Section profiles, material properties and end releases were correctly transferred, however supports, loads and load combinations needed to be input in *FEM-Design*.

7.2.3 Structural analysis

The FE model in the different FEM software was set up so that it matches the model provided by 3D Structural Design as close as possible. This meant that the supports, end releases, loads and load combinations were the same in the model using CAD data as the original model from 3D Structural Design. The structure was modeled with beam elements only and a linear elastic material model for steel. The complete bridge structure in *RFEM* can be seen in Figure 7.22.

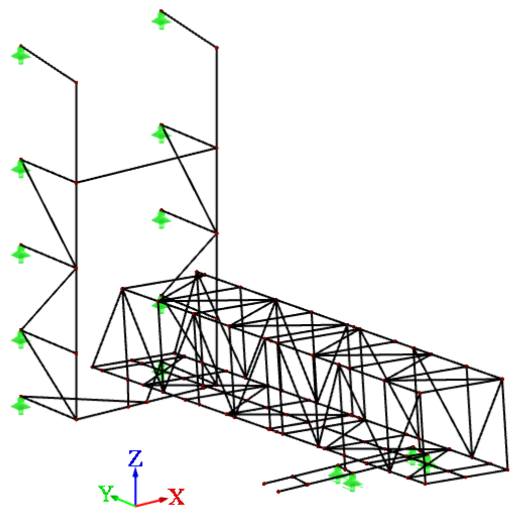
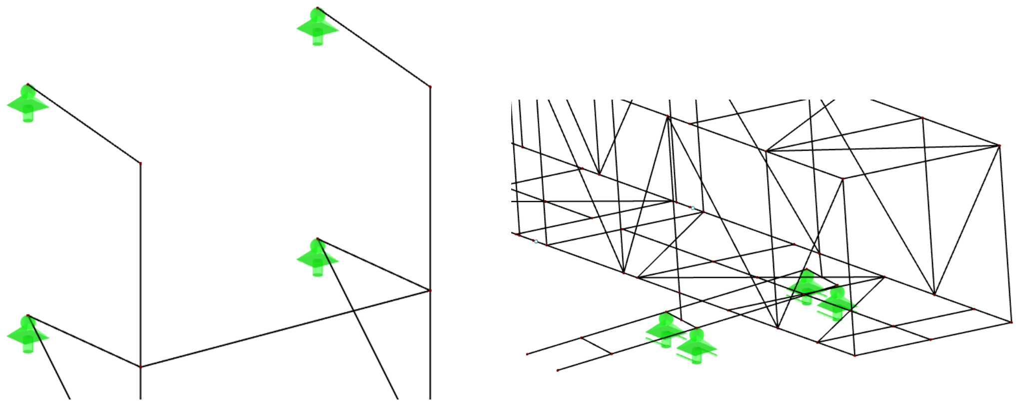


Figure 7.22: FE model of the structure in *RFEM*.

The supports of the connecting frame structure were pinned supports with fixed translational degrees of freedom in the x, y, and z direction, and released rotational degrees of freedom in all three directions. The supports for the bridge were also pinned supports, but the translational degree of freedom in the y direction was also released. The supports can be seen in Figure 7.23.



(a) Frame supports. (b) Bridge supports.

Figure 7.23: Support types of the structure.

Every beam element had a moment resisting connection, except for the connection in the middle of the bridge where there is a hinge in all four connecting chords that only has the translational degrees of freedom released in the beams local x and z direction. Figure 7.24 shows the hinge in the middle of the bridge.

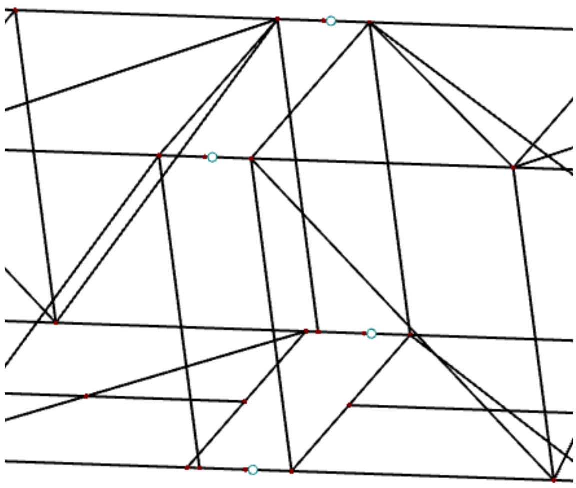


Figure 7.24: FE model of the structure in *RFEM*.

The loads and load combinations were set up based on the *STAAD.Pro* model given by 3D Structural Design. The specific placement and values for the loads and the load combinations can be seen in Appendix DVD *Advanced cases/Conveyor bridge from 3D Structural Design/design_report_conveyor_bridge.pdf*.

After the model was set-up properly a first order, linear analysis was performed. The results of it and the comparison to the original 3D Structural Design model is presented in the following section.

7.2.4 Results and comparison

After the FE analysis was performed in each FEM software the results of them were compared to the original model of 3D Structural Design. For the basis of comparison resultant displacements and support reactions were chosen. The comparison points can be seen in Figure 7.25.

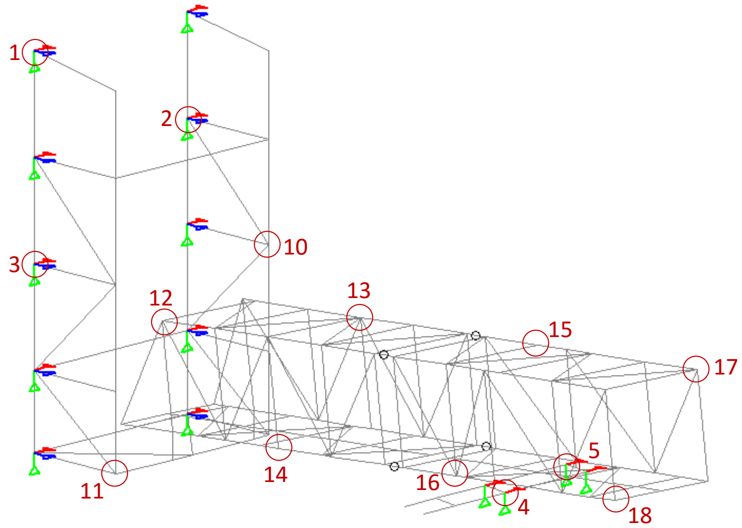


Figure 7.25: Comparison points shown in the *STAAD.Pro* model of 3D Structural Design.

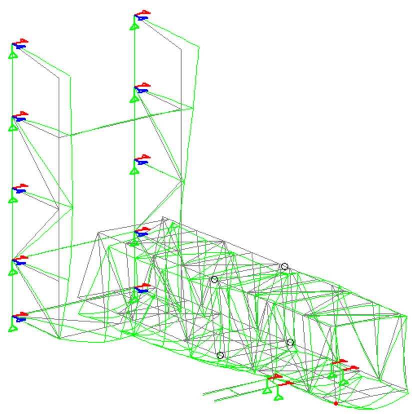
Point numbers under ten are comparison points for support reactions, while point numbers over ten are comparison points for resultant displacements. Naturally, the results for the comparison were taken from the same load combination, namely number 100 c.f. Appendix DVD *Advanced cases/Conveyor bridge from 3D Structural Design/design_ report_ conveyor_ bridge.pdf*. The results are presented in the tables below and the deformed shapes of the structures can be seen in Figure 7.26.

Table 7.6: Support reactions. [kN]

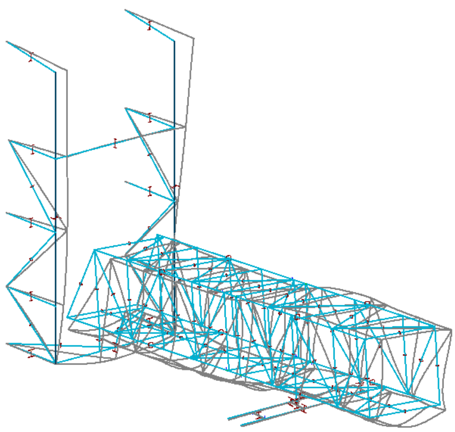
Point	*STAAD.Pro*			*FEM-Design*			*RFEM*		
number	X	Y	Z	X	Y	Z	X	Y	Z
1	2.17	2.27	9.03	2.07	1.19	8.25	2.08	1.68	8.74
2	3.54	116. 57	121.82	3.38	112.64	118.95	3.61	132.64	139.79
3	2.04	0.20	2.76	0.9	1.92	2.72	1.96	0.12	3.17
4	0	6.56	190.06	0	7.36	161.27	0	9.06	189.92
5	0	29.74	213.27	0	28.52	256.92	0	27.10	288.69

Table 7.7: Resultant displacement. [mm]

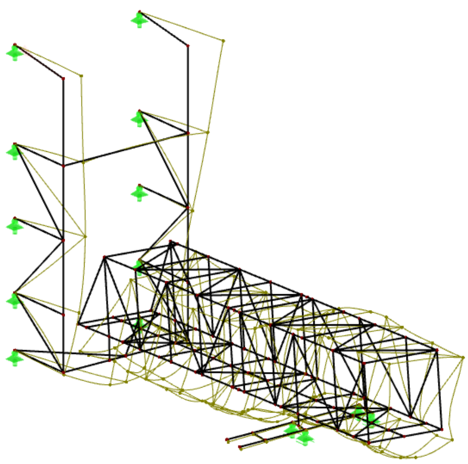
Point number	*STAAD.Pro*	*FEM-Design*	*RFEM*
10	3.31	3.27	3.00
11	0.96	0.86	1.10
12	14.32	14.11	16.28
13	11.98	14.85	16.47
14	12.86	9.79	13.83
15	6.85	12.95	13.01
16	3.84	4.12	5.36
17	5.87	13.11	13.05
18	2.01	5.07	5.73



(a) Deformed shape from *STAAD.Pro*.



(b) Deformed shape from *FEM-Design*.

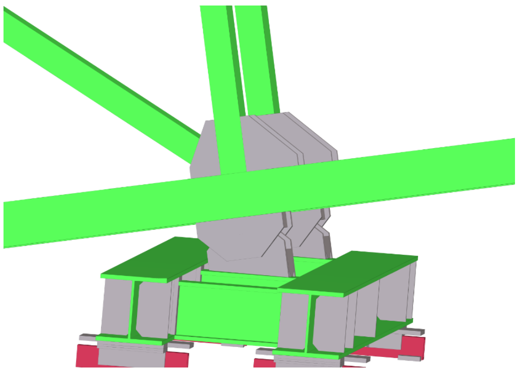


(c) Deformed shape from *RFEM*.

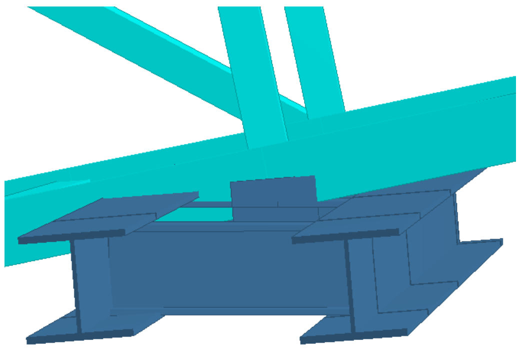
Figure 7.26: Displacements for the different models.

It can be seen from the results that while in general they are close to each other, usually with only a few percent difference, and the deformed shapes follow the reference model fairly well, there are some parts in the structure where the deviation from the *STAAD.Pro* model is quite large. For example looking at points 2, 4, 5 for the reactions and points 10, 11 or 12 for the displacements, the *FEM-Design* and *RFEM* values only slightly differ from the *STAAD.Pro* ones. However, with points 15, 17 and 18 the difference has increased significantly, where for some points it is more than 100%. Although this only happens for small values of displacements and reactions.

These bigger differences occur at the outer part of the bridge that has the connecting support beams. The reason for them is the different geometry that is due to the usage of the CAD model in the creation of the FE model. Figure 7.27 shows the support connection in question. It can be seen that the beam element that connects the bridge to the supporting beams has a shorter length and joins perpendicularly to the beams in the *STAAD.Pro* model, while it is longer and connects at an angle in the *Tekla Structures Learning* model. It is a simplification of the actual connection there, but the accurate geometrical representation in the CAD model leads to a different geometry in the FE model as well.



(a) Bridge support in *Tekla Structures Learning*.



(b) Bridge support in *STAAD.Pro*.

Figure 7.27: Difference in support regions.

7.3 Discussion of general analyses

This section contains conclusions drawn from the general FE analysis of the advanced structures introduced in this chapter, reflecting the effects of incorporating the CAD model in the structural design process.

Similarly to the simple structures, the benefits and drawbacks of using the CAD model is different for the various software and also heavily dependent on the quality of the CAD model itself.

Regarding the model creation in the FEM applications, it only took a fraction of the time compared to the case where it is built up separately. This is a significant advantage, and it is due to the fact that the modeling tools in CAD software are more robust and in most cases easier to use than what is found in FEM applications. As structural geometry is becoming more and more complex the advantage gained from using the advanced modeling tools of CAD software instead of the ones provided by FEM applications is getting more prevalent.

However, a crucial point in structural design is the reliability of the model and the reliability and efficacy of the design process. While the model creation time in the FEM software is greatly reduced, other aspects of the design process are negatively affected by the redistribution of the work load. Such as the preparation of the CAD model for data exchange, and the verification and quality assurance of the FE model.

An additional issue is the transfer of knowledge from one model to the other. The structural engineer has the responsibility to make sure that the FE model is properly created. If he or she chooses to use the data from the CAD model it can pose an issue to verify it. One thing that can help with this issue is if the CAD model comes from a trusted source, where it already passed rigorous quality checks or even better, was created by the same person who creates the FE model. However, that requires on one hand the structural engineer to be proficient in the specific CAD software, and on the other hand a certain redistribution of the tasks associated with structural modeling. Of course the engineer can create his or her own CAD model, based on the one supplied by the architect for example, but this then creates and additional step in the design process, which introduces another potential source for errors.

There are also other areas of benefits of using the CAD model, such as construction management, drawing generation, cost estimates, etc. However, these only present value for companies that also deal with those parts of the design process. If for example a company is only hired as a consultant and their inclusion is limited to load calculation and structural analysis, which often happens, then the integration of the CAD model could only give significant value if it can be used for FE model creation with relative ease and high accuracy. On the other hand for companies who cover a larger portion of the design process a CAD model can deliver several additional benefits aside from it's usage in FE model creation.

Nonetheless, a high quality CAD model is essential for this to work. However, the imported model also needs be thoroughly examined after import, which can take up a significant time. An additional problem with this is that the necessary modifications, which are practically unavoidable, can, in some cases, result in a more time consuming process than if the model was built separately in the FEM software. This reinforces the importance of a properly prepared CAD model. Otherwise, the data exchange would create more problems, and consume more time, when its purpose is to do the opposite.

An important tool that has the possibility to highly enhance the data exchange between CAD and FEM software is the Application Programming Interface (API) of the CAD software. An API is a collection of instructions, tools and rules that define how a software should be built. Many CAD software vendors such as Autodesk, Trimble and Bentley make the API of their respective software publicly available, with an ample amount of support material. This means that basically any user who owns the specific application can use the API to develop his or her own plug-in, extension or separate programme. It can do just one simple task to automate tedious processes or be a full fledged commercial application. Naturally, developing this software means a high initial investment in terms of time and resources, however the potential time saving due to the automation can be very much worth it. As it was mentioned earlier, a major issue with the data exchange was when only the physical model was transferred from the CAD software which resulted in gaps in the analytical model in the FEM application, like in the case of the *Tekla Structures Learning - STAAD.Pro* and the *ProSteel - STAAD.Pro* model transfer. A possible solution to this problem would be to use the API of *Tekla Structures Learning* and *ProSteel* to develop either an extension or a separate program to be able to export the analytical model of the structure, which could then be used for structural analysis directly in the FEM applications.

Chapter 8

Case studies - Advanced analysis

In this chapter two case studies will be presented that are used to examine the capabilities of the data exchange link between *Tekla Structures Learning* and *Abaqus/CAE*. Firstly, the buckling resistance of a frame corner is determined by using a linearised approach, where imperfections and residual stresses are not taken into account. Additionally, a full nonlinear analysis is performed on the same frame corner, where the effects of imperfections, residual stresses and material nonlinearity are accounted for. Finally, a fatigue analysis is carried out on a welded T-joint using the Hot Spot & Notch stress methods.

8.1 Determination of buckling resistance

Failure of a structural component can be categorized in two cases: material failure and structural instability. The focus in the following is structural instability, more precisely the buckling phenomenon. Buckling resistance, R_d is a function of the structural geometry, the material properties, the imperfections and the residual stresses present, according to DNV AS [2013].

To showcase calculations and the data exchange process a frame corner was examined according to an example in DNV AS [2013]. The geometry, material properties and boundary conditions of the frame corner are based on a reference model from DNV AS [2013]. This reference model was a shell model in DNV AS [2013] but in this project aside from the shell model a solid model is also examined. Furthermore, the result of the buckling analysis is also given, which makes it possible to compare the results of the project groups study with the ones from DNV AS [2013].

8.1.1 Model description

The geometry of the frame corner is shown in Figure 8.1.

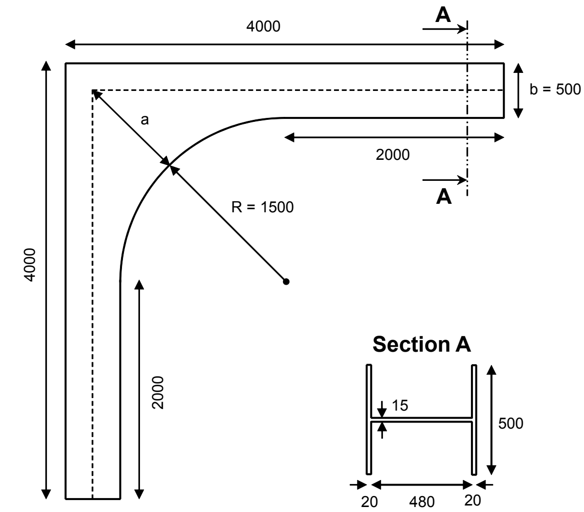


Figure 8.1: Geometry of the frame corner. [DNV AS, 2013]

Based on the geometry above, a steel frame corner was created in *Tekla Structures Learning*. This is shown in Figure 8.2.

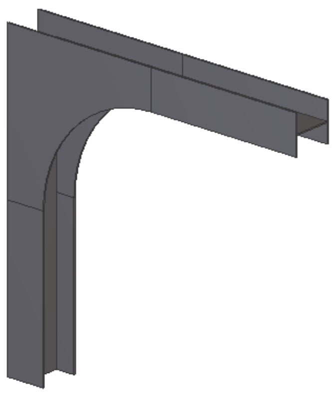
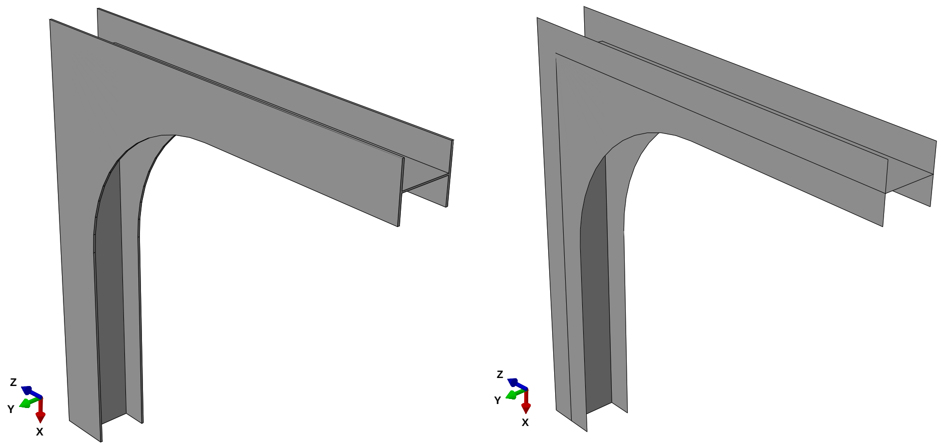


Figure 8.2: Frame corner created in *Tekla Structures Learning*.

8.1.2 Model transfer and encountered issues

First, the *Tekla Structures Learning* model was exported to *Abaqus/CAE* as a 3D solid part. The transfer process is described in Chapter 6 (p. 57), where only geometrical data was transferred. There were no issues connected to the data exchange process. Furthermore, to create the shell model in *Abaqus/CAE* the 3D solid model was converted to a 3D shell model. The procedure to do this can be found in Appendix D.

The solid and shell models are shown in Figure 8.3.



(a) Imported solid model. (b) Converted shell model.

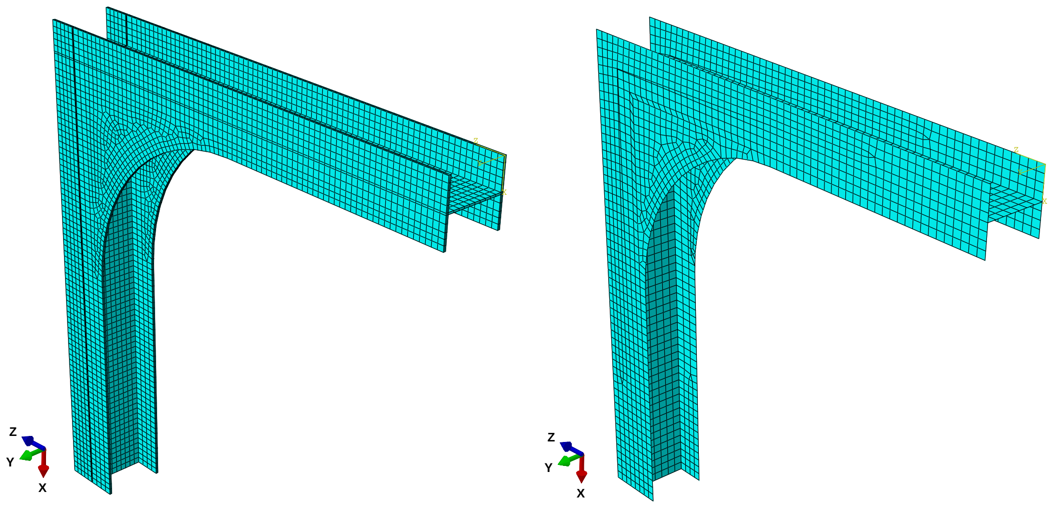
Figure 8.3: Model geometry in *Abaqus/CAE*.

In the linearised approach, the FE method is applied to assess the eigenvalues and maximum von-Mises stresses, which are substituted in empirical formulas to determine the buckling resistance of an ideal linear elastic structure. This method is described in DNV AS [2013], and the study is based on an example from this standard.

Furthermore, the frame was meshed in *Abaqus/CAE* based on the mesh size used in DNV AS [2013]. A convergence analysis was not made by the project group, but the result of a convergence analysis made by DNV AS [2013] was used. However, the meshing of the solid model needed to be finer in order to get an eigenvalue close the one from DNV AS [2013]. It was difficult to mesh the frame structure, especially in the curved part. Several partitions of the frame were necessary in order to have a meshed structure, with no distorted elements. Element size for the shell model was 70 mm, whereas a finer mesh was needed for the solid model with 50 mm.

Element type of S4R, a 4-node, quadrilateral, stress/displacement shell element with reduced integration was used for the shell model. Solid model was meshed with an 8-node, hexahedral, linear brick with reduced integration, C3D8R. C3D8R is a first order element, that only has one integration point which can lead to distorted elements, where the strains calculated at the integration point are all zero. One way to avoid this is to use a fine mesh according to Systémes [2014].

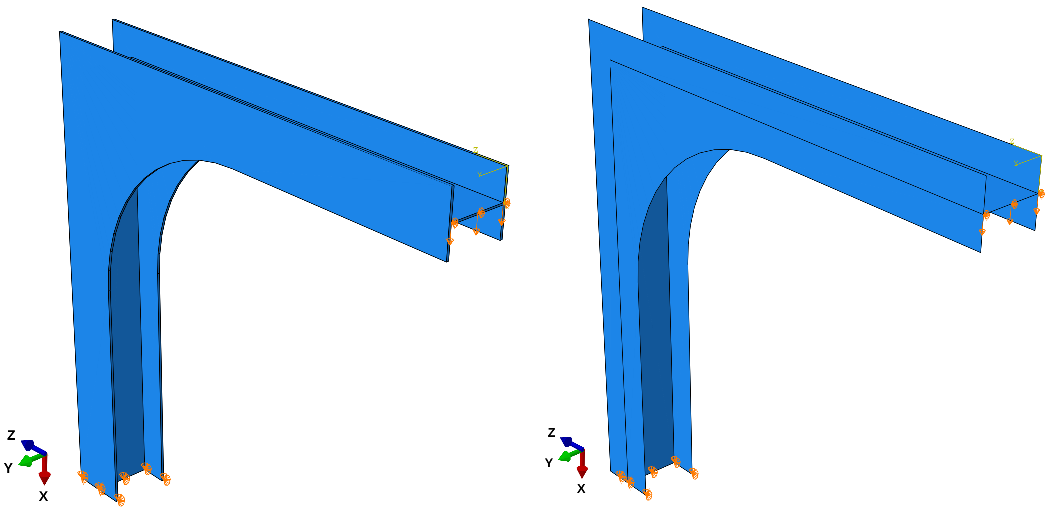
The meshing of the frame corner is illustrated in Figure 8.4.



(a) Meshed solid model. (b) Meshed shell model.

Figure 8.4: Solid and shell mesh *Abaqus/CAE*.

Moreover, boundary conditions were applied to the frame structure corresponding to the reference model.



(a) Boundary condition solid model. (b) Boundary condition shell model.

Figure 8.5: Applied boundary condition in *Abaqus/CAE*.

At the base, the translational degrees of freedom of the frame corner were fixed. At the top, only the translational degrees of freedom in the y- and z-direction were fixed. The loading was applied as a displacement of 10 mm in the x-direction at the top of the frame corresponding to an applied load of $S_{Rep} = 75.7\,\text{kN}$ in the x-direction based on DNV AS [2013].

8.1.3 Linear buckling analysis

Determination of the buckling resistance by use of linearised buckling values is described in several steps in the following c.f. DNV AS [2013]:

1. Build the model. The frame corner is illustrated in Figure 8.3 and the boundary conditions are as in Figure 8.5. The material properties are shown in Table 8.1.

Table 8.1: Material properties

Density	ρ	$7850\,\text{kg/m}^3$
Young's modulus	E	210 GPa
Poisson's ratio	ν	0.3
Yield strength	f_y	355 MPa

2. Perform a general, static analysis for the selected representative load case S_{Rep} showing maximum compressive and von Mises stresses.

3. Determine the buckling eigenvalues and the eigenmodes (buckling modes) by FE analysis.

4. Select the governing buckling mode (usually the lowest buckling mode) and the point for determining the buckling representative stress. The point for reading the representative stress is the point in the model that will first reach yield stress when the structure is loaded to its buckling resistance.

5. Determine the von Mises stress at the point for the representative stress σ_{Rep} from step 2. In *Abaqus/CAE* it is possible to read probe values of stresses, displacements, etc. at certain nodes or elements. In this case this method is used to read the average von Mises stress at a certain node, shown in Figure 8.6. Refer to Table 8.2 for the values of stresses for each model.

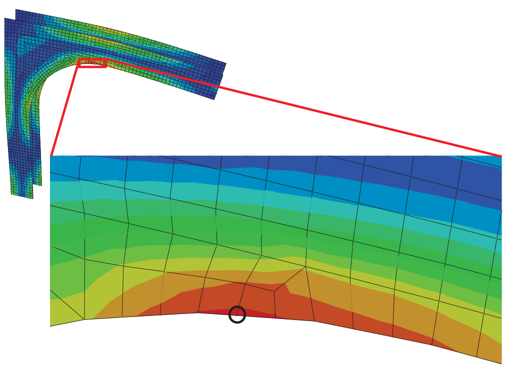


Figure 8.6: Reading of stress values at a chosen node.

6. Determine the critical buckling stress as the eigenvalue for the governing buckling mode times the representative stress.

   The critical buckling stress, σ_{ki} for the governing buckling mode is determined based on the eigenvalue, k_g and von Mises stress at the point of interest σ_{Rep} from step 5. It is given by Equation (8.1):

$$\sigma_{ki} = k_g \sigma_{Rep} \tag{8.1}$$

   The reduced slenderness is given by Equation (8.2).

$$\bar{\lambda} = \sqrt{f_y/\sigma_{ki}} \tag{8.2}$$

7. Select empirically based buckling curve to be used based on the sensitivity of the problem with respect to imperfections, residual stresses and post buckling behaviour.

   Buckling curve, κ for column and stiffened plate and plate without redistribution possibilites is selected, based on Table 5-6 in DNV AS [2013]. The buckling curve is calculated from Equation (8.3).

$$\kappa = \frac{1}{(\phi + \sqrt{\phi^2 - \bar{\lambda}^2})} \leq 1.0 \tag{8.3}$$

$$\phi = 0.5(1 + \alpha(\bar{\lambda} - 0.2) + \bar{\lambda}^2) \tag{8.4}$$

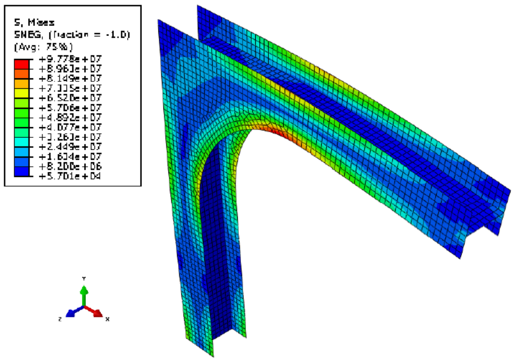
   α is set to 0.3 according to strict tolerances and moderate residual stresses, based on DNV AS [2013]. ϕ is a factor needed to calculate the buckling curve, κ.

8. Determine the buckling Resistance, R_d. γ_M is the material factor, which is set to $\gamma_M = 1.15$ according to DNV AS [2013].

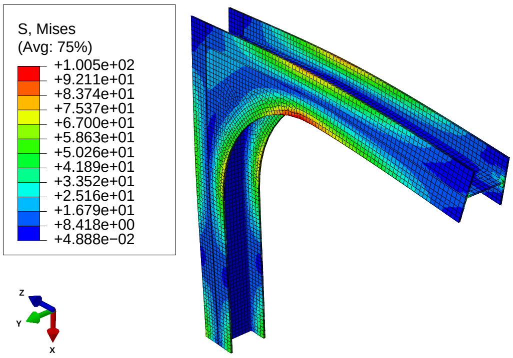
$$R_d = \frac{\kappa f_y S_{Rep}}{\gamma_M \sigma_{Rep}} \tag{8.5}$$

8.1.4 Results and comparison - Linear buckling analysis

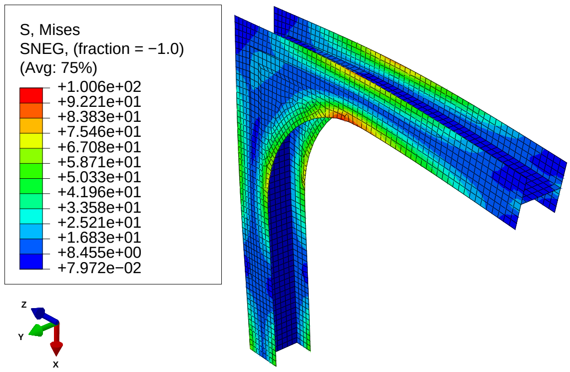
The result of the linear buckling analysis is presented in the following. From the linear analysis only von Mises stresses are shown in the following, since the maximum compressive stresses are not used for further calculations.



(a) Shell model, DNV AS [2013]



(b) Solid model.

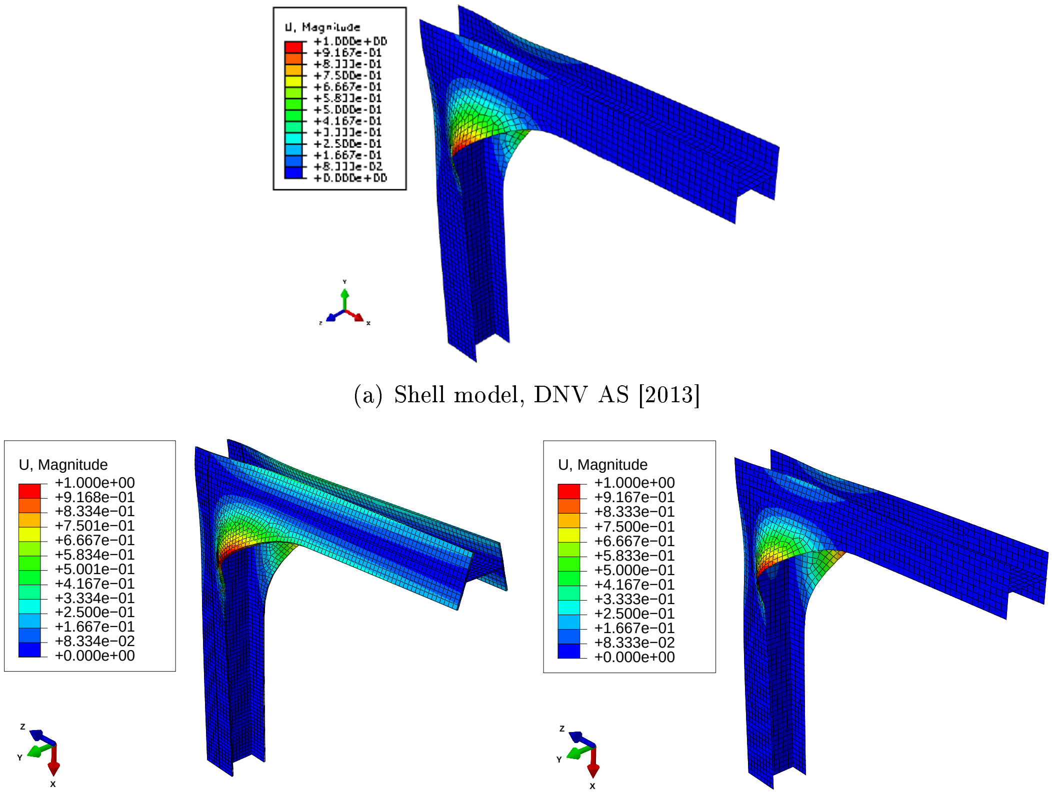


(c) Shell model.

Figure 8.7: Distribution of von Mises stress from linear analysis in *Abaqus/CAE*.

As it can be seen in Figure 8.7 the stress distribution is similar in these three cases. The maximum value of von Mises stresses are presented in Table 8.2.

A buckling analysis is performed in *Abaqus/CAE*, and the first eigenmode from the analysis is shown in Figure 8.8.



(a) Shell model, DNV AS [2013]

(b) Solid model.

(c) Shell model.

Figure 8.8: First buckling mode.

It can be seen in Figure 8.8 the first buckling shape is similar for these three models. Nonetheless, there is a noticeable difference between the solid and shell model, which seems to have more deflection in the upper part compared to the shell models. The eigenvalues are listed in Table 8.2.

In accordance with the example in DNV AS [2013], the first buckling mode is selected. The point for reading the representative stresses are chosen at the point, where the maximum von Mises stress occurs (in this case it is read from a node in the curved part of the frame).

The buckling resistance R_d is calculated based on Equation (8.5) and presented in Table 8.2.

Table 8.2: Results of buckling analysis.

	S_{Rep}	k_g	σ_{ki}	σ_{Rep}	κ	$\bar{\lambda}$	R_d
	[kN]	[-]	[MPa]	[MPa]	[-]	[-]	[kN]
Shell, DNV AS [2013]	75700	6.24	608	97.4	0.77	0.76	184.0
Shell	75700	6.21	592	95.4	0.76	0.77	186.1
Solid	75700	5.72	572	100.1	0.75	0.79	175.5

From Table 8.2 it can be seen that there is a small difference in eigenvalues between the three different models. The noticeable difference is between the solid and shell models. One of the reasons for this deviation is due to the meshing of the model. Changing size and type of elements had a very big impact on the results. The solid model leads to the smallest value of R_d, with a deviation of 4.7 % compared to DNV AS [2013].

Furthermore, the bucking resistance obtained with the shell model is closer to the result from DNV AS [2013]. Since it was less time consuming to run an analysis with the shell model compared to the solid model, it is in this case preferred to work with the model created with shell elements.

8.1.5 Nonlinear buckling analysis

Another approach to calculate the buckling resistance for a structure or part of a structure is a nonlinear buckling analysis. This approach is usually used when the result of the linear buckling analysis is assumed to not give the real response. With this method, the analysis can be extended with taking into account the material nonlinearity, geometrical imperfections and residual stresses.

Since the outcome of nonlinear analyses carry a lot of uncertainty, it is not recommended to apply all the nonlinearities at the same time, because that can lead to a wrong result. Preferably, the application of the geometrical imperfection and material nonlinearity should happen after each other with numerous tries to gain the full control of the model.

Material nonlinearity

Material nonlinearity means when the stress level in an element reaches the yield point the material in that element starts to behave plastically. In plastic behaviour the material follows a nonlinear stress-strain curve. A simplified stress-strain curve can be seen on Figure 8.9, defined by DNV AS [2013] and used in the nonlinear analysis.

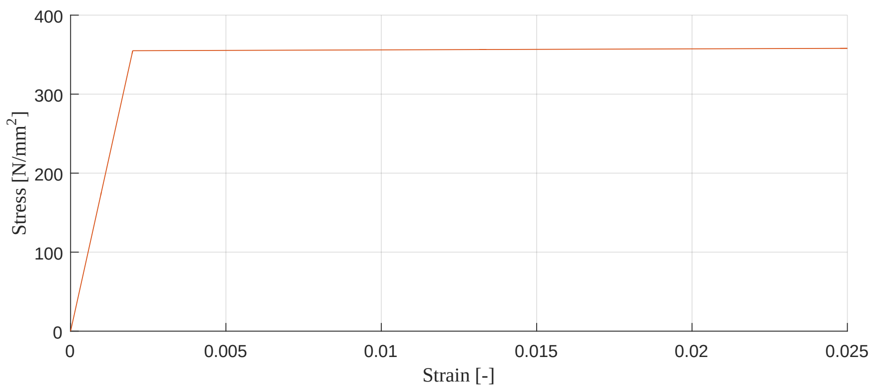


Figure 8.9: Stress-strain curve for structural steel. [DNV AS, 2013]

Geometrical imperfections

In real life, a completely homogeneous material, a perfectly applied axial load and perfect cross sections do not exist. Small imperfections can always be found and these can trigger the buckling in a way that is not expected. Therefore, it is important to take the imperfection into account during the nonlinear analysis. The easiest way to apply geometrical imperfection is to use a small transitional load. Also, it is really hard to estimate the imperfection, therefore predefined values are available in the DNV AS [2013]. For the current case the imperfection was $\delta = 0.0195\,\text{m}$. The calculation of the imperfection can be found in DNV AS [2013].

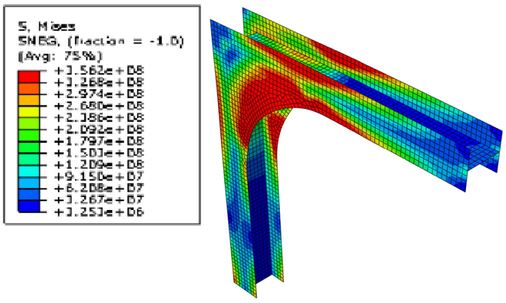
To perform a nonlinear buckling analysis, the same set-up has to be done as in the linear buckling analysis. The geometry, the boundary conditions and the meshing is the same for the solid and the shell model as well. Although, instead of a linear calculation a nonlinear calculation method has to be used, which uses the Arc-length method.

The Arc-Length method is a very efficient method in solving nonlinear systems of equations when the problem under consideration exhibits one or more critical points. In terms of a simple mechanical loading-unloading problem, a critical point could be interpreted as the point at which the loaded body cannot support an increase of the external forces and an instability occurs. [Riks, 1979] This method applies the load incrementally. It is important the choose the load increments with care. The usage of high increments can lead to an incorrect calculation, and *Abaqus/CAE* will not start plasticity calculation.

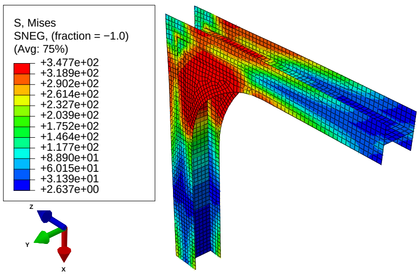
The applied load is the initial load, in this case equivalent displacement, from the linear analysis multiplied with the eigenvalue of the first mode. The nonlinear analysis uses a pattern for the imperfections from the linear analysis, which is the first eigenmode. In buckling the first eigenmode is the most critical for the structure and the analysis takes this pattern as a geometrical imperfection with the defined material nonlinearity.

8.1.6 Results and comparison - Nonlinear buckling analysis

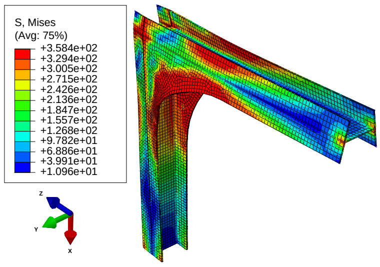
When the analysis is done, the results can be obtained and compared to the results of DNV AS [2013]. Figure 8.10(a) shows the result from [DNV AS, 2013], Figure 8.10(b) and Figure 8.10(c) show the result for the shell and solid model obtained in *Abaqus/CAE*.



(a) Result of the reference model in DNV AS [2013].



(b) Stress distribution of shell elements.



(c) Stress distribution of solid elements.

Figure 8.10: Distribution of von Mises stresses from nonlinear analysis in *Abaqus/CAE*.

It can be seen on the results in Figure 8.10, that the von Mises stress has some minor differences on the two shell models, but on the solid model the stress distribution is not matching the result from the reference model.

Moreover, a force-displacement curve figure was created as a result of the analysis, and compared to the result of DNV AS [2013]. These can be seen in Figure 8.11 and Figure 8.12.

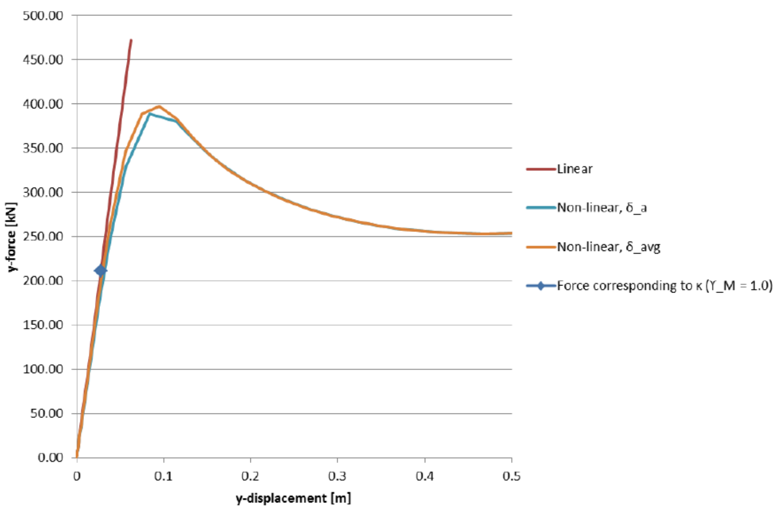


Figure 8.11: Force-displacement curve of the reference model from DNV AS [2013].

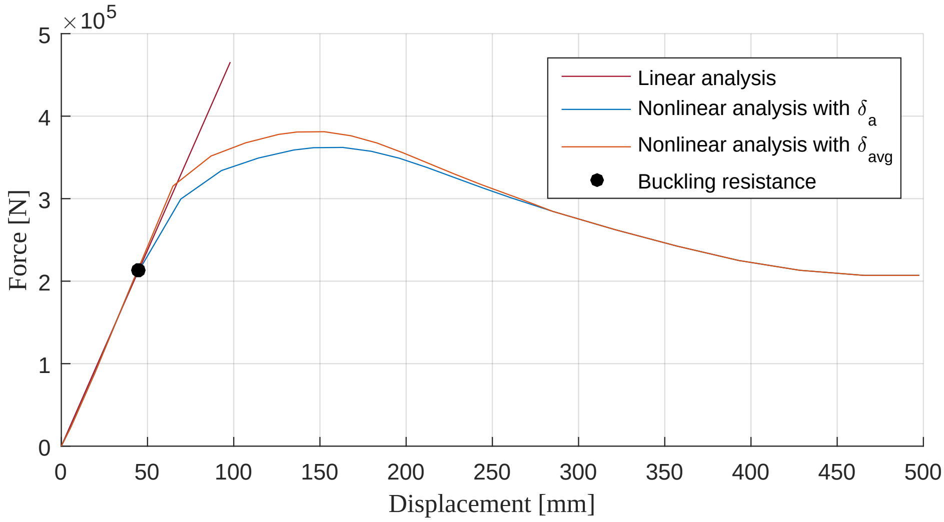


Figure 8.12: Force-displacement curve obtained from *Abaqus/CAE*.

The force-displacement curve also shows some differences. It can be seen on the result of DNV AS [2013] that, the plot converges to 250 KN, but the result of the shell model converges around 200 KN. The reason of this is, during the analysis more increments were used in *Abaqus/CAE* to obtain the closest result.

The usage of *Abaqus/CAE* in the nonlinear buckling analysis was a complex process. As the results are showing previously, two models were used for the analysis, a solid and a shell. For the shell model the result matches - though, with a slight deviation – the result of DNV AS [2013].

For the solid model the case was entirely different. Using the same method and set up as the shell model was built, the results were expected to be the same, but they were not. The stress distribution and the deformation of the beam showed no similarity to the shell model and to the result of DNV AS [2013].

The yielding of the material was also different to the shell model, even though the plastic behaviour of the material was defined the same way for the two cases as DNV AS [2013] defined it.

As the issues were growing, attempts were made to solve the problems and get a closer result to DNV AS [2013]. The first one was to use a finer mesh and modify the element type. The finer mesh, the different element types, like quadratic, did not solve the issues unfortunately, but gave a higher computational time. To reduce this, symmetry was used on the model.

Imperfections are one of the key differences in the nonlinear analysis compared to the linear analysis. Imperfections can be material and geometrical imperfections. While, the material imperfections can be implemented easily, the geometrical imperfections gave some issues during the calculations. Geometrical imperfection can be triggered by small loads or deflections, or initial imperfection of the model can be used. The latter was very promising because, in theory, it is using the nodal deformations from the linear buckling analysis, meaning that the first eigenmode, or the higher modes, depending on the users need, of the linear analysis can be set up as initial geometrical imperfection. As the first eigenmode is the most critical in buckling for a structure, *Abaqus/CAE* can use this pattern as initial starting point, and analyse the buckling phenomena more detailed. Unfortunately, this method did not work, *Abaqus/CAE* did not use the imperfections from the linear analysis, therefore another method was chosen, which was imperfection caused by load or deflection.

Deflection or load defined imperfection has to be applied with care. Concentrated load can give stress singularity at the applied point, therefore displacement defined imperfection was applied, which gave a similar result at the shell model compared to DNV AS [2013].

8.2 Fatigue analysis with Hot Spot & Notch Stress methods

In this section a fatigue analysis of a welded detail will be discussed. This study was chosen to further test the capabilities of the data exchange between *Tekla Structures Learning* and *Abaqus/CAE*. A *Tekla Structures Learning* model of the welded detail was transferred to *Abaqus/CAE* and a fatigue analysis was performed with the Hot Spot & and Notch Stress methods.

Fatigue failure is a failure in the material due to cyclic loading. At a certain load level microscopic cracks can form at places in the structure with high stress concentrations. These cracks can reach a critical size, after which the crack grows at a very high speed resulting in the failure of the structure. [Ralph I. Stephens, 2014] Designing a structure for fatigue can be a very complex procedure depending on structural geometry, loading amplitude, environmental factors, temperature, etc. It also varies based on the type of analysis used, whether it uses numerical data, laboratory testing and whether it employs fracture mechanics or not. A detailed description about fatigue analysis can be found in the recommended practices by Det Norske Veritas (DNV) and the International Institute of Welding (IIW). [DNV AS, 2011] [Hobbacher, 2014] This study only focuses on a few key parts of the fatigue analysis, that are affected by the modeling technique. Fatigue failure is a crucial problem for welded connections thus a simple T-joint with a fillet weld on the front face was chosen to be examined. It has to be noted that the following is first and foremost a study to highlight the aspects of structural modeling, and it should not be taken as a full and comprehensive fatigue design of the T-joint.

8.2.1 Model transfer and encountered issues

The plates of the T-joint are 10 mm thick and the size of the weld is 8 mm. The *Tekla Structures Learning* model of the joint can be seen in Figure 8.13.

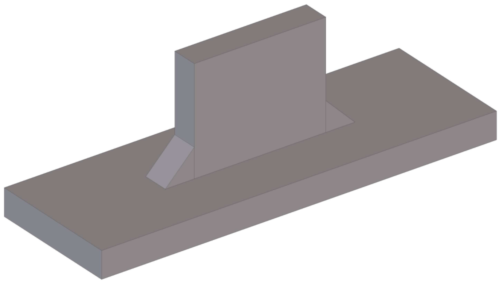


Figure 8.13: T-joint in *Tekla Structures Learning*.

Using the transfer method that was described in Section 8.1 (p. 83) the *Tekla Structures Learning* model was used to create the model in *Abaqus/CAE*. During the import process in *Abaqus/CAE* it was chosen to import the model as a combined geometry with merged solid regions but keeping the intersecting boundaries. This was important because the meshing procedure is greatly simplified this way. Merged solid regions mean that the whole part will be meshed as one single part, while keeping the intersecting boundaries helps setting up a proper meshing pattern. If the intersecting boundaries are not kept at the import the part's geometry becomes too complex for a structured or sweeping meshing algorithm and the bottom-up meshing technique needs to be used, which is a more time consuming method and it can easily be the source of meshing errors and inconsistencies. The transferred model in *Abaqus/CAE* can be seen in Figure 8.14

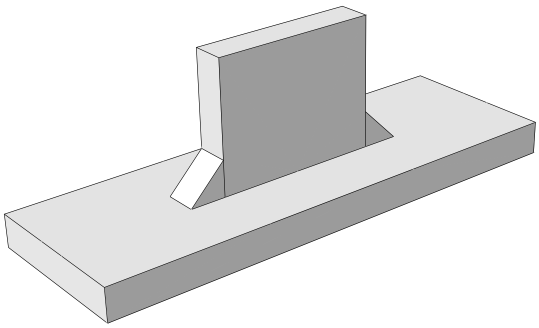


Figure 8.14: T-joint in *Abaqus/CAE*.

There were no issues encountered for this simple model, however for other cases some issues can arise from this model transfer. One problem is connected to *AutoCAD*, and how it creates a 3D solid from the geometrical mesh. For some elements faces of the solid are divided into separate parts, which carries over to *Abaqus/CAE*. This means that these faces need to be combined manually in *Abaqus/CAE* to be able to mesh it properly. This is of course restricted to the case where the model conversion is done in *AutoCAD*.

A further problem with imported parts is that, opposed to parts created in *Abaqus/CAE* the geometry is not editable. This can make it harder to make modifications to the model. Other edit tools like extruding a cut or creating a fillet do work, so many changes can be implemented, but the original geometry can not be modified.

8.2.2 Fatigue analysis

Fatigue life is usually expressed in the number of cycles a structural element can sustain at a certain stress level without failure. For high-cycle fatigue, where the number of cycles is higher than 10^4, an S-N curve is used to determine the fatigue life of the element. An S-N curve, or Wöhler curve, is a logarithmic scaled curve which relates the stress range to the number of cycles before failure. [Ralph I. Stephens, 2014] These curves are determined from laboratory tests on archetype specimens and widely used connection types. Consequently, it depends on material properties, connection type, and the environment among other factors. Design guides and recommended practices like DNV AS [2011] and Hobbacher [2014] specify which S-N curve is to be used for specific cases.

To be able to use these curves and determine the fatigue life of the previously presented structural detail, a stress range is needed. For welded connections fracture usually initiates at either the weld toe or the weld root. Thus a stress range is needed at these locations to assess the fatigue life. There are various methods of calculating the stresses at these points, and from these the Hot Spot & Notch Stress methods were used in the project.

Hot Spot Stress method

The Hot Spot Stress method is used to determine the stress at the certain crucial points in the structure called hot spots. These hot spots are usually at places of sharp corners and transitions, such as a weld toe which is modeled without a radius. To model the weld toe with a radius a very fine mesh is required which increases the computational time significantly. [DNV AS, 2011] However, in many cases there is either no need or not enough resource to use a very fine element mesh, so the weld toe is often modeled without the radius. The stress at the weld toe with a sharp corner in an FE model goes to infinity as the element size approaches zero, thus to determine the stress there, it is first calculated at specified points away from the weld toe and linearly extrapolated to the weld toe. Figure

8.15 shows the extrapolation of the stress to the weld toe, where σ_{HS} is the Hot Spot stress and σ_{Notch} is the Notch stress.

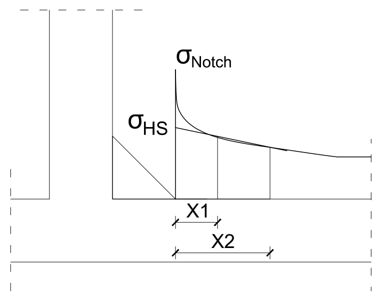


Figure 8.15: Stress extrapolation to the weld toe.

An important part of the Hot Spot Stress method is that it only uses the constant membrane and the linear bending stresses while omitting the non-linear stress peaks. The reason for this is that the effect of the non-linear part is accounted for in the S-N curves. The stress distribution along the plate thickness can be seen in Figure 8.16.

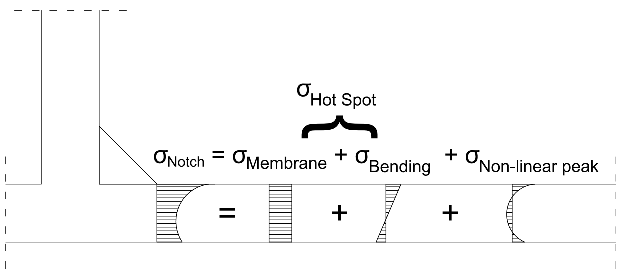


Figure 8.16: Stress distribution along the thickness.

Both DNV and IIW give recommendations for this method which were used and compared in the project. Firstly, they specify the points where the stresses should be calculated. This is given in terms of the distance from the weld toe (X1 and X2 in Figure 8.15). DNV recommends to calculate the stresses at a distance of 0.5 t and 1.5 t from the weld toe, regardless of mesh density, where t is the thickness of the plate where fracture is expected to occur. [DNV AS, 2011] IIW also recommends these values, however only for coarse meshes. For fine meshes it recommends a distance of 0.4 t and 1.0 t. [Hobbacher, 2014] There are also guidelines for additional cases for the places of the stress readouts which can be found in Hobbacher [2014]. In this project the stresses were calculated for both distance pairs.

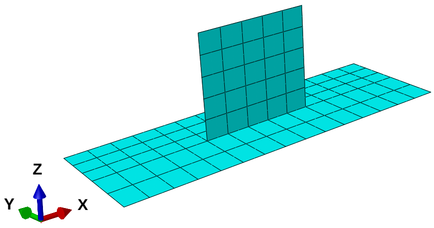
Additionally, these recommended practices give guidelines for the type and size of the mesh that should be used for the analyses. Shell and solid elements can both be used with different criteria. In most cases shell elements are recommended as they can give sufficiently accurate results with significantly lower computational demands than solid elements. An

8-node shell element is recommended with an element size of $t \times t$. This way the stresses can easily be read from the averaged results of the mid-side nodes. It is possible to read the stresses directly because a shell element has a linear stress distribution along the thickness thus the non-linear stress peak is not included. For solid elements a 20-node hexahedral element is recommended with an element size of $t \times t$ and reduced integration. Using reduced integration and only one element through the thickness of the plate results in linear stress distribution which omits the non-linear stress peak again. When using solid elements it is suggested to model the weld profile as well. These recommendations were followed during the testing, however the effect of using a finer mesh with element sizes smaller than $t \times t$ was also examined. The mesh sizes can be seen in Table 8.3.

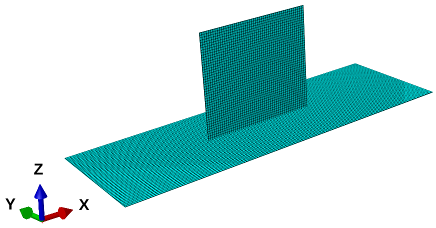
Table 8.3: Mesh sizes

	Coarse	Fine
Shell	10 mm × 10 mm	1 mm × 1 mm
Solid	10 mm × 10 mm	Global: 2 mm × 2 mm | Local: 1 mm × 1 mm

A notable consequence of the finer mesh for the solid model was that there were more than one element in the thickness direction, which can capture the non-linear stress peaks, so the stresses had to be linearised through the thickness. The final meshes of the models can be seen in Figure 8.17 and Figure 8.18.

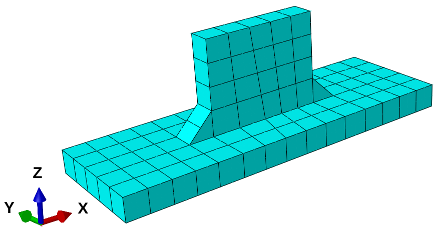


(a) Coarse shell mesh.

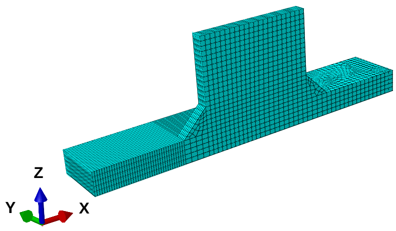


(b) Fine shell mesh.

Figure 8.17: Shell element meshes.



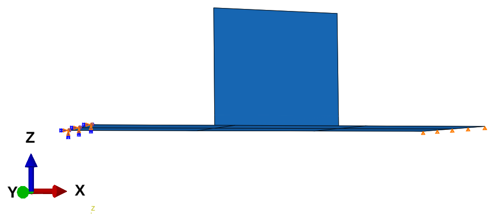
(a) Coarse solid mesh.



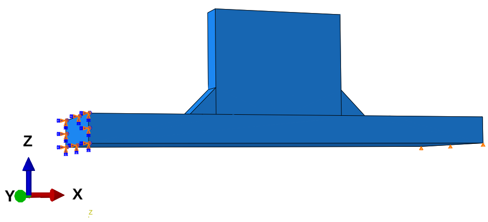
(b) Fine solid mesh.

Figure 8.18: Solid element meshes.

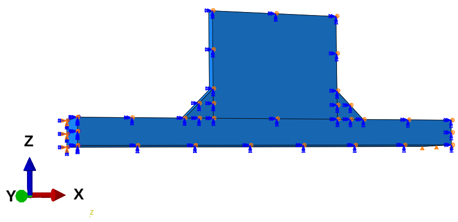
Regarding the boundary conditions all degrees of freedom were fixed on the left edge of the base plate while on the right edge the translational degree of freedom in the z direction was fixed. For the solid model with a fine mesh an additional boundary condition was used as only half of the joint was modeled to save on computational time. For this reason a symmetry boundary condition was implemented on the cut side. These are shown in Figure 8.19.



(a) Shell model boundary conditions.



(b) Solid model boundary conditions with coarse mesh.



(c) Solid model boundary conditions with fine mesh.

Figure 8.19: Boundary conditions.

Choosing a loading that represents the fluctuating loads during the lifetime of the structure is of high importance for fatigue analysis. Every varying load type should be considered, making sure that the load spectrum used during the analysis corresponds to an upper-bound estimate. [Hobbacher, 2014] The main focus of this thesis is not the fatigue analysis itself and for this reason only two simplified load cases were used to calculate the stress range at the weld toe. One was a distributed load of 50 N/mm^2 tension while the other was a 50 N/mm^2 compression. These loads were applied parallel to the x-axis to the right edge of the base plate. The tension case gave the maximum stress σ_{max} while the compression gave the minimum stress σ_{min}. The stress range $\Delta\sigma$ was calculated with

$$\Delta\sigma = \sigma_{max} - \sigma_{min}. \tag{8.6}$$

It also had to be chosen what type of stress should be used at the hot spots. This is affected by the stress state at the weld toe, by the angle between the first principal stress and the normal to the weld profile. [Hobbacher, 2014] In this case the loading was perfectly perpendicular to the weld profile, thus the maximum principal stress was used for the analysis, which had a direction perpendicular to the weld toe as well.

For every model a linear elastic steel material model was used.

The Hot Spot Stress method can only be used to deal with fatigue failure at the weld toe. Cracking from the weld root can not be examined with it.

Notch Stress method

The Notch Stress method is a tool that can be used to assess fatigue failure initiating from the weld toe and weld root as well. It was specifically developed for FEA. A key difference compared to the Hot Spot Stress method is that the weld is modeled with a radius and a very fine mesh is used for the areas in question. [Hobbacher, 2014]. For plates with a thickness higher than 5 mm an effective notch root radius of 1 mm is recommended for the weld toe and weld root as well by IIW. [Hobbacher, 2014] This is shown in Figure 8.20.

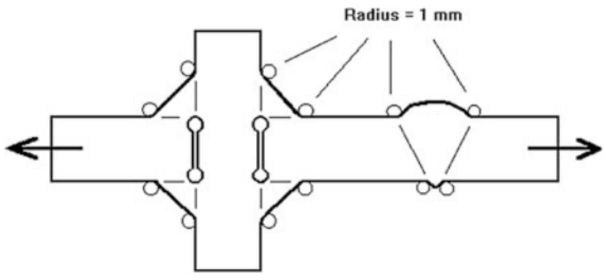
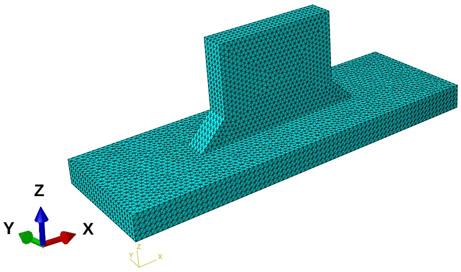


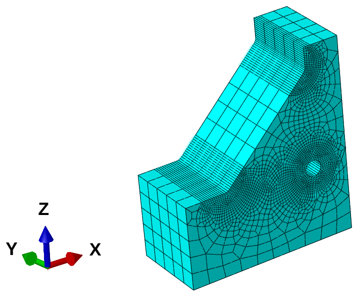
Figure 8.20: Recommended notch radii. [Hobbacher, 2014]

Regarding the model transfer for the Notch Stress Method, the joint was imported to *Abaqus/CAE* without keeping the intersecting boundaries of the solid regions. This was necessary to be able to create the radii between the weld and the plates. A drawback of this is that either the bottom-up meshing tool has to be used for the meshing of hexahedral elements or tetrahedral elements need to be used. In this case the latter was chosen.

As this method requires a very fine mesh it was decided to use the submodeling technique available in *Abaqus/CAE*, without which the computational requirements would be too high. With submodeling the calculation time can be greatly reduced. This technique works by first setting up a global model and performing a global analysis on it. After it is done, a local submodel is created of the parts of the model that need to be detailed. This submodel is connected to the global model via a special type of boundary condition which couples the calculated displacements from the global model to the degrees of freedom on the cut planes. This way only the submodel needs to have a very fine mesh while the global model can have a rather coarse mesh, which keeps it computationally efficient. Based on the recommendations of IIW a seed size of 0.25 mm was used for the notch areas in the local model, and a seed size of 2 mm was used for other parts of the model. [Hobbacher, 2014] The meshes can be seen in Figure 8.21.



(a) Mesh of the global model.



(b) Mesh of the local model.

Figure 8.21: Meshes for the Notch Stress method.

The general boundary conditions, material properties and loading was the same as for the Hot Spot Stress method.

8.2.3 Results and comparison

Hot Spot Stress method

The hot spot stress ranges are presented in Table 8.4, while the stress distribution in front of the weld toe can be seen in Figure 8.22. In the tables and figures below DNV and IIW denote which recommended practice was used for that particular result.

Table 8.4: Stress ranges for Hot Spot Stress method. [N/mm^2]

	Coarse	Fine	
	DNV and IIW	DNV	IIW
Shell	78.08	63.24	66.42
Solid	82.36	70.88	74.67

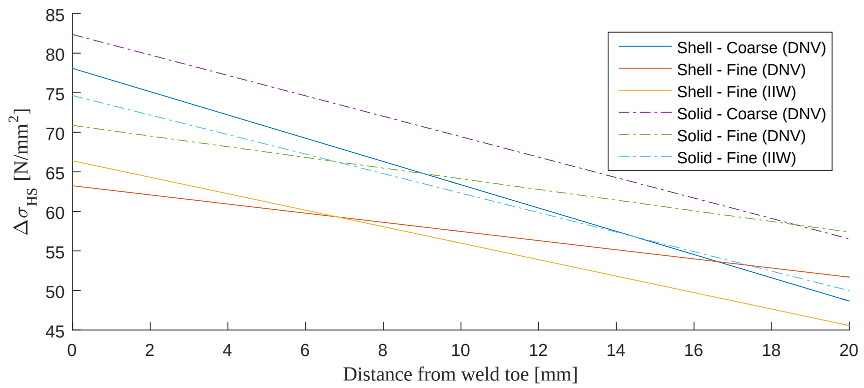


Figure 8.22: Stress distribution in front of the weld toe.

After the stress ranges were determined they were used together with an S-N curve to determine the fatigue life of the weld. The D-curve was used for this as it was recommended by DNV AS [2011]. The S-N curve can be seen in Figure 8.23 while the fatigue lives are shown in Table 8.5.

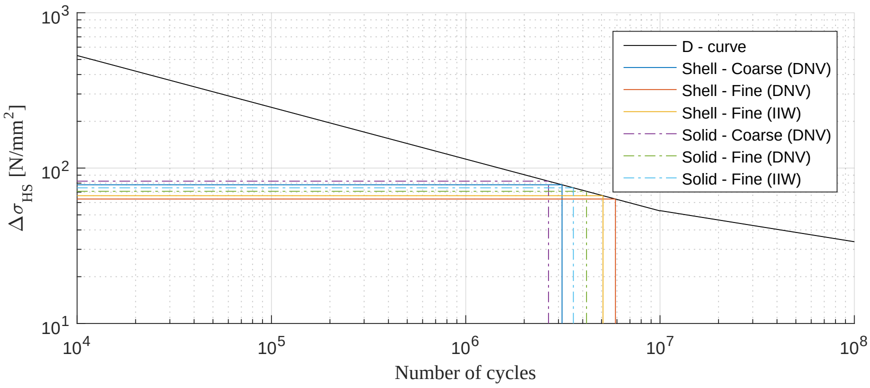
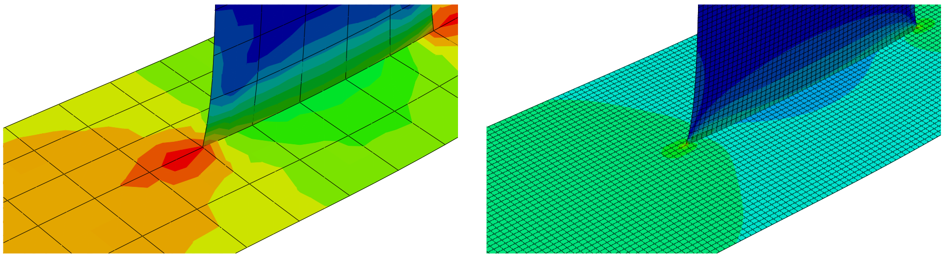


Figure 8.23: S-N curve for Hot Spot Stress method.

Table 8.5: Fatigue life for Hot Spot Stress method. [Cycles]

	Coarse	Fine	
	DNV and IIW	DNV	IIW
Shell	3.1E+06	5.9E+06	5.1E+06
Solid	2.7E+06	4.2E+06	3.6E+06

It can be seen from the results that the element type and mesh size have a noticeable effect on the fatigue life. For example between using a coarse solid mesh and a fine shell mesh the difference can be more than a 100%. The results show that the hot spot stress calculated with solid elements tend to be higher than for shell elements. Coarse meshes for both element types also produce higher hot spot stresses than the fine meshes. The reason for this is that with a high mesh density close to the discontinuity, in this case the the weld toe, the effect of the discontinuity is more localised, i.e the stress concentration only affects a smaller area closer to the weld toe itself. This results in lower stresses away from the weld toe. This is shown in Figure 8.24.



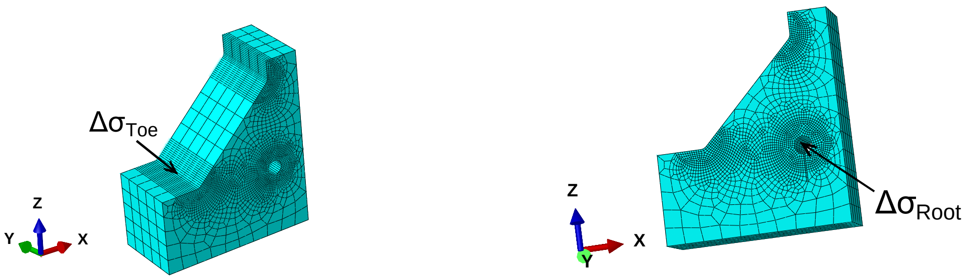
(a) Coarse shell model. (b) Fine shell model.

Figure 8.24: Stress distribution at the weld toe.

Consequently, the lower stresses correspond to higher fatigue lives, which in turn shows that the recommended coarser meshes by DNV AS [2011] and Hobbacher [2014] are giving a conservative result. Coarser meshes give results on the safe side in terms of fatigue life, however they can also lead to overdesigned structures. This highlights a drawback of this method, namely, that the post-processing phase, i.e. how the stresses can be read at the specified positions influences the decisions made in the meshing part of the preprocessing phase.

Notch Stress Method

Figure 8.25 shows the places of the stress read-outs, while Table 8.6 shows the stress ranges of the Notch Stress method.



(a) Stress readout place for the toe. (b) Stress readout place for the root.

Figure 8.25: Stress readout places.

Table 8.6: Stress ranges of the Notch Stress method. [N/mm^2]

$\Delta\sigma_{\text{Toe}}$	$\Delta\sigma_{\text{Root}}$
158.71	126.26

IIW recommends the FAT 225 S-N curve to be used together with the Notch Stress method, which can be seen in Figure 8.26. [Hobbacher, 2014] This was used to calculate the fatigue life for both the weld toe and the weld root. These are presented in Table 8.7.

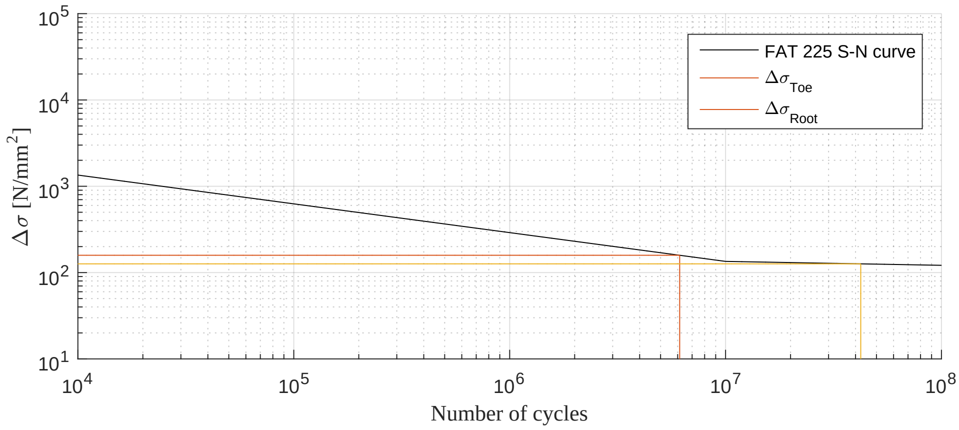


Figure 8.26: S-N curve for Notch Stress method.

Table 8.7: Fatigue life for the Notch Stress method. [Cycles]

Toe	Root
6.1E+06	4.2E+07

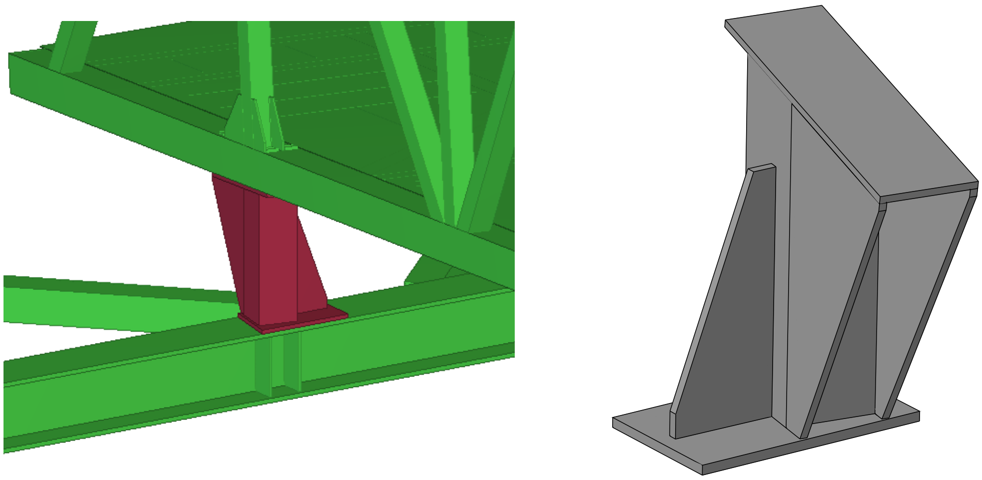
Comparing the results from the Notch Stress method to the results of the Hot Spot Stress method it is obvious that the Notch Stress method captures the stress peak at the notch which results ina significantly higher stress. Looking at the fatigue life the difference is less noticeable for the fine hot spot shell meshes but is still significant for the other models, especially for the coarse ones. This seems to reinforce the notion that using the specified mesh sizes in DNV AS [2011] and Hobbacher [2014] can underestimate the fatigue life of a welded detail.

8.3 Discussion of advanced analyses

This section will contain a discussion about the case studies made in this chapter. It reflects the findings of the project group regarding the use of CAD models for advanced FEA.

Using CAD data for the model creation in *Abaqus/CAE* has definite advantages. Even though *Abaqus/CAE* has a very capable modeling toolset, being able to skip the modeling phase can save a noticeable amount of time, especially for complex model geometries. Naturally, the biggest benefit comes for the case where solid elements would be used for the analysis. It is also advantageous for shell elements, but in that case the solid elements need to be transferred to shell elements. This requires additional time and modification, but it could still be easier and faster to do than building a model entirely in *Abaqus/CAE*.

A further advantage of the transfer method is that the position of each part in 3D space is also carried over. For instance when importing a detail with multiple connecting parts as a single part, which can be seen in Figure 8.27 it can save valuable time that the location of each element is automatically imported so the assembly is ready for use and no manual positioning is required.



(a) Bridge support in *Tekla Structures Learning*. (b) Bridge support in *Abaqus/CAE*.

Figure 8.27: Bridge support model transfer.

For cases where small parts with connecting sides need to be modeled, such as weld profiles, additional applications could be helpful to make the modeling easier. Creating the exact weld profile, for example, can be done in both *Tekla Structures Learning* and *Abaqus/CAE*, however it is potentially much easily done in dedicated CAD software for mechanical engineering purposes, like *SolidWorks* or *Catia V5*. Even though *Tekla Structures Learning* has advanced features to model welds, those are focused on the weld properties and they are not modeled as physical objects. On the other hand mechanical engineering CAD software do focus on the physical weld profiles, which can be extremely useful for cases where the FE analysis needs to be done on the welded detail. An additional benefit of *SolidWorks* or *Catia V5* would be that they have a direct link with *Abaqus/CAE*, which makes the data transfer faster and easier.

Chapter 9

Summary of interviews

The project group conducted interviews with several engineering companies from Denmark, including Rambøll, COWI, MOE and 3D Structural Design. The purpose of the interviews was to get a picture of the level of BIM implementation in the design process in each of these companies.

9.1 Interview with COWI

This interview was conducted with Jens Kristian Lund Birkmose over phone from the Odense office of COWI. Jens is a structural engineer and BIM Process Specialist at COWI. His responsibilities includes optimizing the use of *Revit* and other Autodesk products through content and standards development.

Structural engineers at COWI use CAD and FEM software in the design process. They try to match those CAD software that architects use, since the architects may already have been working for a year or more with a project, when the engineers come into the design phase. CAD and FEM software used in the building department in COWI are listed in Table 9.1, and the choice of the software depends on the type of project and what they have to deliver to the client.

Table 9.1: Software used at COWI.

CAD	FEM	Hand calculation
Revit	*RSA*	*Excel*
Microstation	*Abaqus/CAE/Ansys*	*Mathcad*
Tekla Structures		

Often, during the design process a combination of *Excel* and *RSA* is used, for example *RSA* for steel framing and *Excel* for instability calculation and prefabricated design. *Abaqus/CAE* or *Ansys* are used for foundation design only. Furthermore, for bridge projects software that is built in house is used. Detailed design is done by the manufacturers.

In COWI the FE model is built up in the FEM software from the ground up, nothing is imported from the CAD software. The purpose of the 3D model the architects create is to show how the finished structure will look like, whereas the structural engineers use the 3D model to show how to build the structure. The problem is that people who create the CAD model in e.g. *Revit* do not have the know-how about how the FEM software e.g. *RSA* worked. The structural engineers who know how the model works in the FEM software, do not know how to model in CAD software. According to Jens, the ideal scenario would be if the structural engineers knew how CAD modeling works and the draft persons had knowledge and a general understanding of FEM. Furthermore, Jens thinks that the structural engineers need to build the CAD model as well, to have the correct analytical model for calculation. This is already implemented in Norway, where they have no designers, only engineers. They do both the drawings and also the calculations. COWI in Denmark is not there yet, but they are moving forward in this direction.

Moreover, Jens argues that even importing the geometry from CAD software is not useful, because the structural engineer has not created the model and may spend 80 % of the time figuring out why the models looks like it does and trying to change that. Whereas if the model is created from scratch in the FEM software 80 % of the time can be used to optimize the model instead.

To communicate with the architect during the design phase, PDF format is used. They are looking at a new format called BCF (BIM Collaboration Format), which is a way of communicating design changes and comments. This is not implemented yet. IFC is not used either, unless the client demands it.

The worst developed aspect of the structural design process is not a technical problem, but the phase where the architects change the design a lot which means recalculating and keeping track of changes. It is important to keep track of changes, otherwise a situation might arise where they don't know what has changed. If the integration between the CAD and FEM software becomes better, the recalculation will be easier by keeping track of changes during the design process.

9.2 Interview with Rambøll

The interview was conducted with Anders Bilgaard and Morten Dalsgaard at Rambølls Aalborg office. Morten Dalsgaard is head of the building department, coming from a position as structural engineer. Anders Bilgaard is a constructing architect ("bygningskonstruktør" in danish), and he is primarily working with BIM software. Furthermore, Anders is a member of a group at Rambøll, that develops certain items in *Revit* like templates, families, etc.

In the design process, as CAD software primarily *Tekla Structures* and *Revit* is used. *Tekla Structures* is used for structural projects in the power plants department and sometimes in the building department if the structure is very detailed. Whereas *Revit* is used for regular projects in the building department, because it has both a structural part, and a mechanical part for installations. FEM software are not used at all in simple projects, where *Excel*, *MathCad* etc. is used. A list of different software used in the building department in the Aalborg office can be seen in Table 9.3. Detailing (e.g. details of steel connections, etc.) are often done separately in simple calculation spreadsheets.

Table 9.2: Software used at Rambøll.

CAD	FEM	Hand calculation
Revit	*RSA*	*Excel*
Tekla Structures		*Mathcad*

In small projects the structural engineers build the model up in their FEM software from scratch. If the geometry is very complicated, then the model is built up in *Revit* and exported to FEM software. Data exchange between CAD and FEM software is only used if it saves time. So the decision whether or not to use BIM depends on how complicated the geometry is. Moreover, the structural engineers are usually more expensive than construction architects, therefore a structure with a complicated design is created by construction architects in CAD software.

Primarily centrelines and other geometrical data is used from CAD model for FE analysis. Sometimes, but not often, more detailed export is used from *Tekla Structures* to *RSA*, where profile data is transferred as well. Material parameters are often handled in the FEM model, and if necessary implemented in the CAD model. According to Anders, the most time consuming part of the data exchange process is to create a useful calculation model. The CAD model often has a different purpose than the calculation model, and should represent the actual built structure which is often very different from the model that can be used in FEM software. Furthermore, the process of going back and forth many times before the final result is time consuming.

Issues when importing the geometry is often that centerlines of structural members do no meet each other, which leads to cross-sectional eccentricity. All the members must be in line, otherwise a lot of unwanted bending moment etc. will be in the model due to small eccentricities in the nodes or near the nodes.

Rambøll plans to increase the use of BIM in the future. According to Morten, it is about the right mindset. The older generation of engineers (with 30 + years of experience) prefer to design by hand calculation, which is ok for non complex buildings. So there is a lot of tradition that needs to be challenged. Young engineers are more capable to adjust and more used to changes.

The worst developed aspect of the structural design process are the high number of changes. Changes in the early phases require a lot of time both from the engineers and architects.

Morten and Anders would like a software, that would make the communication with the architects easier. The architects should be more clear about their ideas and the engineers could explain their challenges.

9.3 Interview with MOE

The interview was conducted with Mads Vangsgaard at the Aalborg office of MOE. Mads is head of department at MOE in Aalborg.

At MOE in Aalborg, the structural engineers use a combination of *Revit* and *Excel* regularly. This work flow is rather new at the Aalborg office, before hand calculations were used. Excel spreadsheets are made based on Eurocodes.

The following CAD and FEM software are used at MOE. *Tekla Structures* is not used in the Aalborg office, but is in the Copenhagen office.

Table 9.3: Software used at MOE.

CAD	FEM	Hand calculation
Revit	*RSA*	*Excel*
Tekla Structures	*Finwood*	*Mathcad*
AutoCAD		

Revit and *RSA* are used separately. MOE at Aalborg had no succes with *Revit* and *RSA* integration for now, so data exchange between CAD and FEM software is not used. Mads also points out that architects should not add loads on the structure, this is where structural engineers come in. However, in case MOE gives information about loads on e.g. a slab to a company who designs the slab, it is relevant to add the loads in the *Revit* model. This is done for prefabricated concrete elements, where the structural engineer gives the load to a company that designs the element. The company designing the element gains a lot by optimization, the structural engineer does not gain anything by optimizing the element.

According to Mads the worst developed part of the structural design process is the changes the architects make. It is time consuming. If the architects did not introduce a lot of changes, the process would be more economically beneficial.

9.4 Interview with 3D Structural Design

This interview was conducted with Knud Hjortflod Nielsen at 3D Structural Design office in Hjørring. Knud is a structural engineer and owner of the consulting engineering company.

Knud has worked with 3D modeling for almost 20 years now, including structural calculations and structural design. At 3D Structural Design, they always build a 3D model, even if it is a simple structure, either in CAD or FEM software, very often in both. The software listed in Table 9.4 are used in the design process.

Table 9.4: Software used at 3D Structural Design.

CAD	FEM	Hand calculation
Revit	*STAAD.Pro*	*Mathcad*
Tekla Structures	*SAP2000*	
	RFEM	
	RSTAB	

When it comes to the level of detail in the models, it depends on what is going to be delivered to the client in the end. It could be just the amount of steel needed for a structure or production of the entire set of structural drawings, structural documentation, etc. They either start with a 3D CAD model or FEM model and can transfer the model either way with CIS/2 format. If it is a simple structure, where they are only calculating more or less roughly, they often start with the FEM software. Whereas, if exact 3D modeling is produced by them afterwards, it is easier to have all basic general arrangement drawings fixed before they start on the calculation.

The model is at the most 50% finished, when the data exchange between CAD and FEM software stops. "*Once detailing starts, the connection is lost*". Sometimes no data exchange is done between CAD and FEM software, and the models are built in parallel but separately. However, there is communication between the two people creating the models (sometimes just one person is responsible for both CAD and FEM modeling). About data exchange between CAD and FEM software, Knud points out, that there is also a bit of doubt. Does it work, or does it not work? And in that case, it is better to have two separate models and control exactly what is happening in the CAD and FEM software.

At 3D Structural Design, they prefer to build up their own CAD and FEM model. If the model is coming from other companies, they create a reference model in the FEM software and export it to *Tekla Structures* to check if the overall geometry is OK. If they receive the model in IFC format, they can use the model as if it was originally created in *Tekla Structures*. Otherwise, it gives more difficulties than it actually helps, and they don't use the model at all.

When it comes to the detailing of standard joints, RFEM modules for joints are used, that are fully integrated with extracting the section forces from the calculation and put them in the joint design modules. For non-standard joints, hand calculation is done with *Mathcad*.

One of the issues about implementing BIM in the building industry is that the different parties in a project do not use the same software. Different software interfaces and databases are used. Another issue is that, e.g. the structural engineers and the contractors are not at the same stage. Sometime the structural engineers at 3D Structural Design need information about e.g. type of an elevator in the construction, in order to do detailing around the elevator. But the people responsible for the elevator do not know this at the right time or change the elevator type later in the process. Revision takes a lot of time, and it costs time and money. There are a number of revisions during the process of a project, who should manage this?

"The amount of data is not the issue anymore. The issue is the administration of the databases and making sure who is responsible for each stage and who is approving at each stage."

Chapter 10

Conclusion

In this chapter a conclusion is presented which incorporates the findings from the case studies and interviews and presents answers for the problem formulations devised in Section 1.4 (p. 8).

How can the implementation of BIM in the design process help structural design?

Models with complex geometries can be significantly easier to transfer from a CAD software than to create in a FEM application from the ground up. This requires a properly set up CAD model and a well functioning exchange link. A key point of the data exchange for practical viability is its ability to save time during structural design. However, it should be noted that finding a way to not only transfer data but knowledge as well is fundamental for an efficient BIM integration.

Feedback from structural engineers can also be easily given to other participants in the project. This in turn enables that the results from the structural analysis can be implemented faster and easier. As in many cases building design is a highly iterative process, this has the possibility to decrease time and consequently costs of the design.

Which method of data exchange has the highest potential and which direction should the field of data exchange be developed?

Based on the results of the case studies it was found that the direct links between applications performed the best both in terms of technical capabilities and ease of use. Both the *Revit - RSA* and *Tekla Structures Learning - RFEM* links showed great potential as they were able to transfer every information necessary for structural analysis in an easy and fast way. The StruXML file format was also very capable and a good alternative even though it had some issues e.g. property mapping limitations. The CIS/2 file format, while being available for only steel structures, also proved to be a valid option for data exchange since it could transfer the required information in a simple way. On the other hand, the IFC based data exchange methods experienced severe problems and constrictions compared to direct links. The transferred data was mostly limited to geometry and certain section properties. It was also only able to deal with the physical model and not the analytical representation which would have been needed for structural analysis. The ISM format performed very poorly in the case studies. Reliably only geometry could be exchanged while material and section properties could only be transferred in specific cases. Finally, the data exchange link between *Tekla Structures Learning* and *Abaqus/CAE* proved to be capable of transferring geometrical data properly, however the requirement for the additional step in the process using *AutoCAD* is far from ideal.

While it is evident that the proprietary direct links performed better than the open-source IFC method, the project group strongly believes that future research and developments in this topic should be focused on open-source methods rather than proprietary ones. Even though direct links work really well in a closed system, data exchange can be severely hindered in projects where multiple disciplines have to work together using software from different vendors. Open-source solutions like IFC on the other hand have the potential to provide a way for data exchange in interdisciplinary environments. One obstacle in front of a better utilization of IFC and the reason for its slow adoption is the fact that software vendors need to work towards and commit resources to provide and develop IFC integration for their own products. However, this might not be in their best interest because a well developed IFC integration could be a viable competitor for the direct links. This would mean that engineers are not limited to the specific vendor's applications and can choose other products while still being able to use data exchange just as efficiently as a direct link.

Most CAD software vendors allow for the free use of their software's API, which makes it possible for engineering companies to develop their own plug-in, extension or standalone programme for data exchange. Naturally, this requires software development expertise from the companies side, and an allocation of resources and time for this task. But considering the possible value it can bring in the future in the form of a faster and more efficient design process it can be a worth vile investment.

During the case studies the project group experienced numerous problems with the section and material property recognition between various CAD and FEM applications. It is the project groups belief that a common standard or agreement among software vendors that all adhere to can greatly reduce this issue. It should specify one common method for each application to use for its section and material library so property mapping can become significantly easier.

The project group suggests that the modifying the current principles of the data exchange processes can be considered by software developers and project managers at engineering companies. At the time of this writing the way data exchange is carried out is by first creating an analytical representation of the physical model in the CAD software and then transferring that to FEM applications. The problem with this approach is that in general the person who creates the CAD model does not have knowledge about FEM. However, for a successful data exchange the CAD model should be built up and prepared based on considerations for FEM. One possible solution for this could be to move the creation of the analytical model from the CAD software to the FEM software. The result of this would be that the structural engineer has direct control and an easy way to set up the FE model based on his expertise and tailored to his needs. Another solution, connected to the assignment of roles and task within a project, can be to redistribute the task of creating the CAD model to the engineers. This way the knowledge is also preserved because only one person deals with both the CAD and FE models. Obviously, this would require high level CAD expertise from engineers.

How can this interoperability be implemented in engineering practice?

BIM has proved to be a useful technology decades earlier but it has still not been fully implemented in the structural engineering discipline. One of the reasons for this is that the design process and workflow needs to change and adapt to a BIM based one, and there is a strong resistance against this change from many engineers who insist on sticking with the old design methods. However, leaving the old design processes behind for more advanced ones that incorporate BIM is essential for it to be successful in the structural engineering discipline. Structural engineers need to be convinced of the value BIM integration can give. A key point for this is to streamline the process as much as possible and making sure that modifications in the models during data exchange can be easily monitored and controlled.

Due to the integrated workflow and data exchange BIM enables, communication between parties involved in the project is of paramount importance. Constant discussion and a clear agreement on how to conduct the modeling and who is responsible for certain tasks is indispensable for a successful project.

In order to have a well functioning data exchange, the appropriate software needs to be chosen. As it was presented in the case studies some applications work better together than others and have features that may have been overlooked in the past but have become more relevant in recent years. Companies need to take a look at their current contracts with software vendors and need to make sure that their software portfolio is well suited for their needs, and if necessary, make changes so they are properly equipped to handle projects that require a high level of BIM integration.

Chapter 11

Reference Lists

Bibliography

Advanced Solutions, Inc. (2016). *BIM Yields Significant Benefits*. URL : http://www.advancedsolutions.com/design/services/lifecycle-bim.aspx. 27-02-2016.

American Institute of Steel Construction (2016). *CIS/2*. URL : https://www.aisc.org/content.aspx?id=26044. 27-04-2016.

Autodesk (2014). *Integrating Autodesk Revit, Revit Structure, and Robot Structural Analysis Professional*. User Guide. Autodesk.

Autodesk Inc. (2016a). *Dynamic Structure Analysis Mode*. URL : https://knowledge.autodesk.com/support/robot-structural-analysis-products/learn-explore/caas/CloudHelp/cloudhelp/2015/ENU/Robot/files/GUID-6020B158-0F05-4754-B69E-2ABDEB610110-htm.html. 27-03-2016.

Autodesk Inc. (2016b). *Elasto-plastic Parameters (Section)*. URL : https://knowledge.autodesk.com/support/robot-structural-analysis-products/learn-explore/caas/CloudHelp/cloudhelp/2015/ENU/Robot/files/GUID-AC349646-B20F-43D5-A4D3-25085D6BC282-htm.html. 27-03-2016.

Autodesk Inc. (2016c). *General (Columns / Beams - EC2)*. URL : https://knowledge.autodesk.com/support/robot-structural-analysis-products/learn-explore/caas/CloudHelp/cloudhelp/2015/ENU/Robot/files/GUID-310E4A0B-FF86-4D9B-9FD4-667971F9F5A1-htm.html. 27-03-2016.

Autodesk Inc. (2016d). *Mesh Generation Options (Solid Modeler)*. URL : https://knowledge.autodesk.com/support/robot-structural-analysis-products/learn-explore/caas/CloudHelp/cloudhelp/2015/ENU/Robot/files/GUID-C5070E0E-EA34-424E-A4C6-91D6E6A879D0-htm.html. 27-03-2016.

Autodesk Inc. (2016e). *Non-linear Static Analysis.* URL : https://knowledge.autodesk.com/support/robot-structural-analysis-products/learn-explore/caas/CloudHelp/cloudhelp/2015/ENU/Robot/files/GUID-FB3C86D3-0E30-43A6-82D6-6C50F429FA0D-htm.html.
27-03-2016.

Autodesk Inc. (2016f). *Powerful structural equation modeling.* URL : http://www.autodesk.com/products/robot-structural-analysis/features/code-based-verification/list-view.
27-03-2016.

Bentley (2009). *Bentley Communities Structural - Wiki.* URL : http://communities.bentley.com/products/structural/w/. 08-02-2016.

Bentley Systems, Inc. (2016a). *3D Structural Analysis and Design Software.* URL : https://www.bentley.com/en/products/product-line/structural-analysis-software/staadpro.
27-03-2016.

Bentley Systems, Inc. (2016b). *STAAD.Pro.* URL : http://goo.gl/fb6kHJ. 27-03-2016.

Dlubal Engineering Software (2010). *Interface RX-Tekla.* User Guide. Dlubal Engineering Software.

DNV AS (2011). *Fatigue Design of Offshore Steel Structures.* Recommended Practice. Det Norske Veritas AS.

DNV AS (2013). *Determination of Structural Capacity by Non-linear FE analysis Methods.* Recommended Practice. Det Norske Veritas AS.

C. Eastman, et al. (2011). *BIM Handbook.* Handbook. John Wiley and Sons Inc.

A. Grilo & R. Jardim-Goncalves (2010). 'Value proposition on interoperability of BIM and collaborative working environments'. *Automation in Construction* **19**(5):522 – 530. Building Information Modeling and Collaborative Working Environments.

K. Z. Gustavsen (2012). 'Interoperability between architectural BIM models and structural analysis software'. Master's thesis, Technical University of Denmark, Brovej, Bygning 118 2800 Kongens Lyngby.

T. Hansen (2011). 'Udvikling af IFC-kompatibelt dimensioneringsprogram'. Master's thesis, Technical University of Denmark, Brovej, Bygning 118 2800 Kongens Lyngby.

A. Hobbacher (2014). *Recommendations for Fatigue Design of Welded Joints and Components.* IIW Collection. Springer International Publishing.

I. Kovacic, et al. (2014). *Assessment of BIM Potentials in Interdisciplinary Planning through Student Experiment and Practical Case Study.* CORP – Competence Center of Urban and Regional Planning.

T. Liebich (2013). *IFC4 – the new buildingSMART Standard.*

Model Support Group of the IAI (2007). *Industry Foundation Classes IFC2x Edition 3 Technical Corrigendum 1.* URL : http://www.buildingsmart-tech.org/ifc/IFC2x3/TC1/html/ifcstructuralanalysisdomain/lexical/ifcstructuralloadgroup.htm. 26-02-2016.

A. K. Nielsen & S. Madsen (2010). 'Structural modelling and analysis using BIM tools'. Master's thesis, Aalbor University, Sohngaardsholmsvej 57, DK-9000, Aalborg.

L. Oberwinter, et al. (2013). *Interdisciplinary, BIM-supported Planning Process.* Diamond Congress Ltd.

L. Qin, et al. (2011). 'Industry foundation classes based integration of architectural design and structural analysis'. *Journal of Shanghai Jiaotong University (Science)* **16**(1):83–90.

T. Racz & T. Olofsson (2009). *Interoperability challenges of an engineering software provider*, pp. 275–285. CRC Press, Taylor & Francis Group.

R. R. S. H. O. F. Ralph I. Stephens, Ali Fatemi (2014). *Metal Fatigue in Engineering.* as. John Wiley & Sons, Inc.

E. Riks (1979). 'An incremental approach to the solution of snapping and buckling problems'. *International Journal of Solids and Structures* **15**(7):529 – 551.

Spatial Corp. (2016). *3D ACIS Modeller.* URL : http://web.archive.org/web/20070309194852/http://www.spatial.com/products/acis.html, Note = 03-06-2016.

StruSoft (2015a). *StruSoft StruXML Revit Add-In, Manual: Using Revit – FEM-Design link.* User Guide. StruSoft.

StruSoft (2015b). *Tekla StruXML Export, User's Guide to Tekla Structures - FEM-Design Integration.* User Guide. StruSoft.

StruSoft AB (2016a). *FEM-Design.* URL : http://goo.gl/UPrqgf. 27-03-2016.

StruSoft AB (2016b). *FEM-Design Applied Theory and Design.* URL : download.strusoft.com/FEM-Design/inst110x/theory.pdf. 27-03-2016.

D. Systémes (2014). *Abaqus 6.14, Abaqus/CAE User's Guide.* User Guide. Dassault Systémes.