INTEGRATED DESIGN SYSTEMS

HAMK
HÄMEEN AMMATTIKORKEAKOULU
HÄME UNIVERSITY OF APPLIED SCIENCES

Bachelor's thesis

Visamäki, Construction Engineering

Spring 2019

Mariia Ovanesova

HAMK
HÄMEEN AMMATTIKORKEAKOULU
HÄME UNIVERSITY OF APPLIED SCIENCES

Degree Programme in Construction Engineering
Visamäki

Author	Mariia Ovanesova	**Year** 2019
Subject	Integrated design systems	
Supervisor(s)	Cristina Tirteu (HAMK UAS), Ilari Pirhonen (AINS Group)	

ABSTRACT

The purpose of this Bachelor`s thesis was to conduct an investigation of integrated design systems implemented in the structural design. Since the topic is large, the focus of the research was on data exchange between Tekla Structures and RFEM. For the start, the investigation about integrated design was held to familiarize with the topic, outline the reasons to use such systems and define the main problems making the application of the software connecting tools risky and unstable. Three methods were considered to adjust the integration – "Direct data exchange" option in RFEM, "Analysis and Design" model" option in Tekla Structures, and separate program called "Rhinoceros" with add-on module called "Grasshopper". It was concluded that the last method is the most advantageous. Moreover, since the research was conducted under the supervision of the company AINS Group, the existing tools developed by the company to set connections between software (Tekla Structures, Grasshopper and RFEM) was allowed to be used.

During the practical part, the data exchange between the programs was done using the example model represented by a steel frame. The original structure was modelled in Tekla. Then, all elements from Tekla as one input were transferred to Grasshopper, where the necessary modifications were completed. In order to bring the modifications to the automatic level, additional tools were developed as Grasshopper components assigning buckling lengths, creating rigid links, converting profile names etc. The tools were created in Grasshopper with the help of algorithms. The received model was imported to RFEM, where it appeared with correct properties.

In conclusion, the approach of integrated design simplifies the structural analysis and geometry modelling. The connection of models from Tekla Structures and RFEM via Rhinoceros with Grasshopper gave the most appropriate results among other methods. However, this method still should be improved and checked in real projects.

Keywords   Integration, RFEM, Tekla Structures, Grasshopper, Rhinoceros

Pages   31 pages

CONTENTS

1 INTRODUCTION

The use of IDDS (Integrated Design & Delivery Solutions) "involves changes in each of the project phases from conceptual planning and business case formulation to all stages of the supply chain: design, construction, commissioning, operation, retrofit and decommissioning" (Prins & Owen, 2010, p. 4). In fact, the data integration is implemented at all stages of construction process because all the phases are connected with each other. For example, the air gap in one of the walls is increased by an architect. First, the change will appear on the architectural drawings. Then, the engineering models and drawings will be modified. Gradually, all the project documents, calculations, ordered elements and so on, affected by that change will be edited. The procedure is conducted manually for most of the changes at any stage. With the help of an automatization, the process can be accelerated, and the mistakes due to the human factor can be reduced. That is why the integration by the software should be developed.

In the designing process of the construction a wide range of different factors and parameters is considered: environment, purpose of use, surroundings, budget etc. In addition, there are values which should be set depending on initially given factors. For example, the sizes of profiles in the load-bearing columns are depended on the applying loads (among other factors). Moreover, while the design process is going on, almost all factors and values can be changed due to different reasons: significant changes in the project or timetable, lack of money, new data appearing and so on. The computer-aided designing programs allow to modify the design quite fast. However, the process of turning computer into the main designing tool was stated relatively recently. Therefore, the ways of data collaboration are still under development. Basically, such data collaborations or automatic changings of the model due to the change of several values represent the parametric designing method. The changed values can also be called parameters.

The method followed by Parametric Architecture can be summarized as follows: definition of the term, identification of the parameters involved with the theme, and definition of the analytical relationships between dimensions dependent on the various parameters (Bucci & Mulazzani, 2002). However, it can also be applied in structural design. For instance, the bolted connection with gusset plates is the term or object to design. The plate thickness and the number of bolts can be defined as parameters. The key values affecting those parameters are forces acting in the connection.

Even though the Integrated design approach covers all the construction phases, in this study, the part of structural design was considered. As a matter of fact, nowadays the large variation of software is available for structural engineers. Commonly, the separate programs are used for 2D-modelling, 3D-modelling and model analysis. Therefore, several of them are involved in each project. In reality, the connection between the programs works with plenty of mistakes and information losses or is not existing at all. Consequently, the modelling of the same structure should be completed separately in each of utilized programs. Thus, the proper connection of the programs is demanded to

be created. That is why the primary purpose of the study is to investigate the available methods for integration between structural software.

2   INTEGRATED DESIGN

Even though Integrated Design approach is an innovative and developing field, the system had already been used at the end of the 19th century by Antonio Gaudi. He was the Spanish architect creating works in his own style. Because beside the imagination, he had implemented new approaches in order to make better balanced structures. Some of his ideas are described in the next chapter.

Definitely, integration was applied already a long time ago by structural engineers. However, the methods are always updating due to the rising level of technical and software abilities. Nowadays, due to the application of the integrated design in a wide range of spheres, the improving of the system is going at high speed. Visiting the seminars concentrated on the theme is one way to receive the latest news. Therefore, the seminar about Rhinoceros and Grasshopper organized on 24th January 2019 in Aalto University was attended. The main innovations affecting structural design is written in Chapter 2.2 named "Rhinoceros User Meeting".

The other remarkable sign of the importance of the approach importance is that it is used in many projects across the world. The structures vary a great deal in terms of material, size, purpose of use etc. The method is flexible and has a high potential in the engineering field; hence, it becomes more and more common there.

2.1   Sagrada Familia

As was mentioned above, Antoni Gaudi was one the first who used integrated design in Architecture and Building design. Mainly, new methods were applied while designing the Sagrada Familia between 1883 and 1926. The structure became quite unique and uncommon, so even nowadays, during the construction phase, the newest tools have to be used or even further developed to make the construction of the church possible. Moreover, another difficulty for the engineers is a tight timetable. According to the plans all works should be completed in 2026 (Fundació Junta Constructora del Temple Expiatori de la Sagrada Família, 2018).

The remarkable innovation in the designing processes was the method to determine the correct structure using the free-curved surface. Antoni Gaudy made the hanging chain model with ropes and weights (on the left in Figure 1). Subjected to continuous loads of its own weights, the ropes formed the catenary curves between connection points. Hence, the minimum energy state system was set up naturally (Makert & Alves, 2016). Then due to the "Hooke´s law" the members under tension in the upside-down placed model became elements under pure compression in the real building. The method continues to be developed even now. For example, the roof of Kitagata Community Center was created using the same logic but already not as a real model but as a

parametric model by architect Arata Isozaki. "Sensitivity Analysis" proposal was used instead of chains (on the left in Figure 1). Consequently, the modification of the shape with updated analyses of the strain energy´s transitions takes several minutes (Sakamoto & Ferré, 2008).

Figure 1 On the left - model with ropes and weights (Dragicevic, 2018), on the right - Kigata Community Center (davidaewen, 2015)

In addition, Gaudi made the parametrically designed columns. The process is shown in the Figure 2. Initially, two columns are rotated around the central lines in opposite directions. Then, the received shapes are connected into one column. Finally, only those parts are left which were present in both columns. Such method is used for all the columns in Sagrada Familia, where those have a varying diameter and height. The rotation angle is in direct proportion to the size of the column. That means that the bigger the column is, the smaller rotation angle it has (Hernandez, 2006).
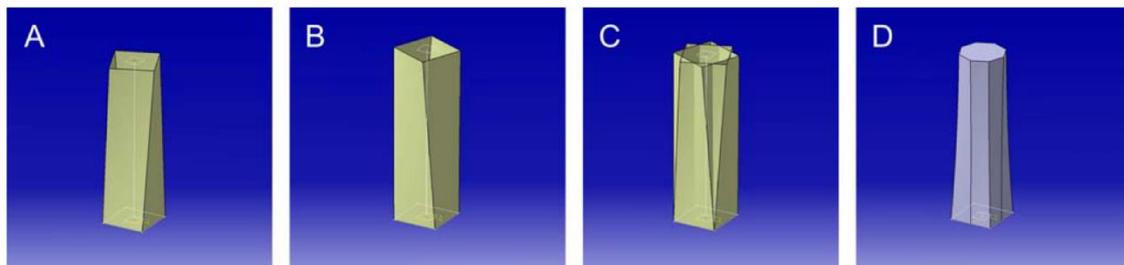

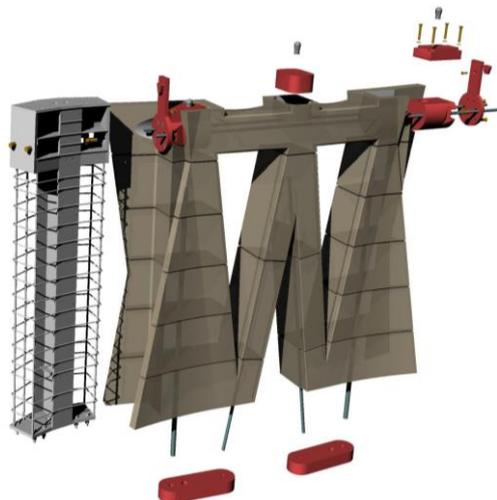Figure 2 Parametric design of columns (Hernandez, 2006, p. 317)

Figure 3 Tensioned-stone panels (Fundació Junta Constructora del Temple Expiatori de la Sagrada Família, 2018)

As a result of the architectural innovations, it was demanded to make an update of the structural design methods. For instance, the tensioned-stone panels were developed for the six central towers of the church. The element is uncommon, because it combines stone and reinforced concrete (see Figure 3). Additionally, the shapes were designed similarly to the described above parametrical columns. In order to produce reinforcement for the columns, the algorithms in Grasshopper were set which automatically generate all those panels with reinforcement in Tekla Structures (Carrasco, 2019). As a result, the new structure became easy to design and modify, and time which is strictly limited was saved.

All in all, the updated concepts helped not just to create a new type of the structure but enforced further development of the construction methods. Above only two innovations of Antonio Gaudi are described. However, it should be mentioned that he has done more in this field. Moreover, there were also Luigi Moretti, Frei Otto and others working in this direction. In fact, the implementation of parametric approaches leads to the new challenges in structural design. However, the development of integrated design part related to the engineering software is the decision of those problems.

2.2   Rhinoceros User Meeting

One of the possible methods to set up the integration is to implement the Rhinoceros and Grasshopper (see Chapter 3.4 "Integration via Rhinoceros and Grasshopper"). One advantage for using these programs is that the users are actively communicating with each other. Even more, several times a year, the company "McNeel Europe" (producer of the example software) organizes user meetings for everybody who is interested in parametric and integrated design. In 2019, one of such meetings was held at Aalto University on 24th January. The meeting had two main areas – the fair of the companies and the lectures. At the fair companies developing software for structural engineers were presented among others. During the lectures, the new tools or completed projects were described. In the next paragraphs of the Chapter, some of the innovations are described based on information received at the parametric design meeting (McNeel, 2018).

It should be emphasized that the topics related to structural engineering field had a significant part during the meeting. One of the examples is an engineering company CT Ingenieros which had been using the parametric design approach while working on the new airport of Santiago de Chile. The method was applied because there were four terminals with a similar structure. As a result, the designs for the first, second, third and fourth terminals were completed in 6 months, 4 months, 1 week and 26 hours respectively. As it was concluded by the lecturer, even though it took more time for the first part, then, time was saved while designing the next three terminals.

Another feature, which is becoming possible with integrated design (particularly, with Rhinoceros and Grasshopper), is the creation of virtual reality for a 3D-model. That means that any structure modelled in three-dimensional space (using the programs such as Revit or Tekla) can become an object of virtual reality without excessive modifications. The technology was shown by companies named "Robert McNeel &

Associates", "Mindesk", "Varjo" and "Enscape". In general, the idea is to use virtual reality (VR) as one of the main designing modes in addition to 2D models and 3D models. It is already possible to proceed design work in VR mode as it would be the standard interface.

To conclude, the event was informative. The sufficient familiarization with the technology and idea was received. The lectures all together introduced the tendencies and plans in the integration design. Since the field was not studied precisely before by the research author, the understanding of the current purposes in the field helped to concretize the targets for this study.

3   INTEGRATION BETWEEN TEKLA STRUCTURES AND RFEM

3.1   Structural software

Among all the available structural designing software, Tekla Structures and RFEM were chosen for the research. Consequently, the existing data exchange tools connecting those programs were investigated. Then, additional tools were created as a solution for some problems of existing system.

Tekla Structures is a software for the 3D-modelling of the multi-material structures (Trimble Solutions Corporation, 2018). The main feature of the program is to create structures consisting of elements with defined properties as names, profiles, materials and so on. Tekla Structures is made by Trimble Solutions Corporation which was set up in 1978 (Trimble Inc., 2018). There are plenty of guides and support contacts on "Tekla Campus" web site (Trimble Solutions Corporation, 2019).

RFEM is a program for structural analysis based on modular software system. The program is used to define structures and materials, to add loads for planar and spatial structural systems containing plates, walls, shells and members. The creation of combined structures, model solid and contact elements is also one of the RFEM features. (Dlubal Software GmbH, 2019.) The program is made by Dlubal Engineering Software. The company was set up in 1987. Its developers pay attention to the creation of user-friendly interface. So, the software does not demand a lot of time to familiarize with it. (Dlubal Software Engineering, 2009.)

As a result, the programs are used for different purposes in structural modelling. At the same time, the models in both programs should have the same basic geometry to keep analysis correct in RFEM and create suitable elements in Tekla Structures. Usually now the models are developed separately in the programs. Therefore, the main elements should be completed twice to get them in both places.

At any project stage, there are usually several modifications done to the structure. That leads to editing of all the affected data. The integration with the software brings those modifications onto an automatic level, excluding mistakes due to the human factor and reducing the time spent by a worker to complete that process.

3.2   Direct data exchange between Tekla Structures and RFEM

In purpose to connect models in Tekla Structures and in RFEM, Dlubal Engineering Software corporation has developed the data exchange options (see Figure 4). The tool allows to export models from Tekla Structures to RFEM and reimport cross-sections and internal forces. Another opportunity is a direct exchange of the physic model between the programs. In addition, data exchange of the physical models is also possible (Dlubal Engineering Software, 2010). All three types do not require additional software. In this study, the direct import from Tekla Structures to RFEM is examined excluding the export.
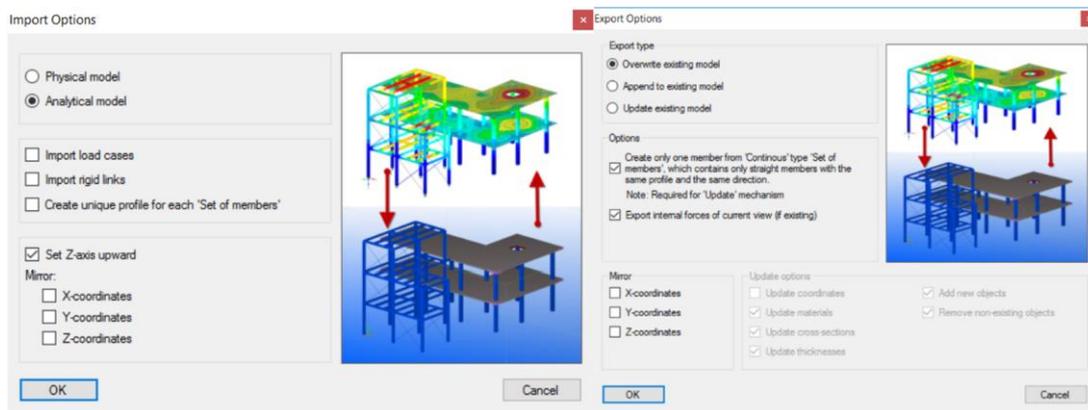

Figure 4 Import and export options in RFEM

In fact, the import from Tekla to RFEM causes plenty of mistakes in the models after transferring (see Figure 5). For example, some of the profiles and the materials have different names in the programs. In such cases, the elements are transferred to RFEM as lines without member properties (as black lines on the right model in Figure *5*). The next problem is that the eccentricities are not considered properly (see Chapter 4.2 "Offset-settings"). In addition, the effective lengths (see Chapter 4.1 "Effective length") are not specified. The data exchange tool itself is not functioning properly. Even though it is possible to make some of the sufficient geometry modifications as division of elements into segments at intersections with other elements, the available tools for editing the model while exchange are quite limited. So, it is not possible to set the correct member types, effective lengths or offsets. For those problems the possible solutions are also proposed in Chapter 4 "Developed tools".
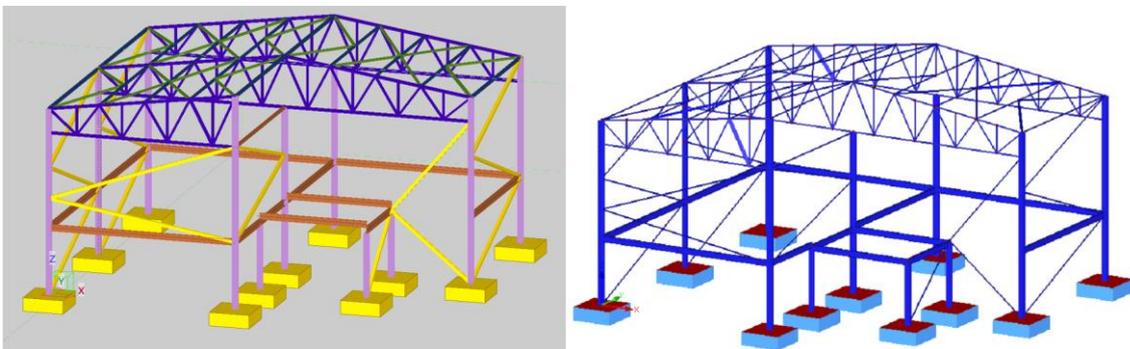

Figure 5 On the left - the Tekla model created manually, on the right - the RFEM model created by direct data exchange option

Due to obvious omissions of the direct link, the usage of it brings more additional challenges than advantages. However, the well-working connection is able to significantly enforce the designing process. Moreover, it makes possible to design the complex structures economically and fast. Therefore, the independent way with possibilities of development should be found.

3.3 Analysis model in Tekla Structures

It is clearly seen that the direct data exchange system described in the previous chapter has many faults. Therefore, additional tools correcting or preventing the mistakes in the generated model should be applied. To be prepared for transferring from Tekla to RFEM, the physical model (structural 3D model) can be turned into an analysis model. The analysis model is used on one hand for analyzing structural behavior and load bearing, and on the other hand for design (Trimble Solutions Corporation, 2016).

In Figure 6, the two cases are presented– the physical model (on the left) and the analysis model (on the right). The second model contains the parts created by Tekla – bars, members and areas of the physical parts. The nodes and support conditions for them are also added. In places of small eccentricities, the rigid links, which are elements whose end points are not moving in relation to points on opposite ends, are placed. It is also possible to add the loads and combinations of them. After all settings are done, the structural calculations from Tekla can be obtained. Then, the analysis model with the loads separately from the physical model can be brought to RFEM via direct data exchange module (see 3.2).
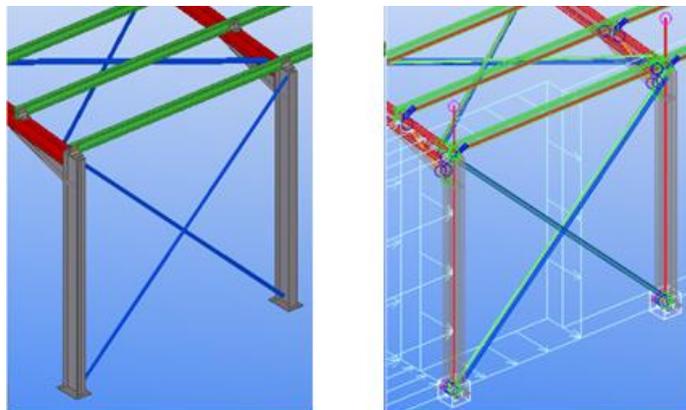

Figure 6 physical (on the left) and analysis (on the right) modes (Trimble Solutions Corporation, 2016, pp. 8-9)

In general, the analysis and design model is a useful feature in Tekla Structures. However, based on the real experience (Pirhonen, 2019), the system contains mistakes, which are described in further in this chapter. One of the serious consequences of it happened in the project, where it was decided to use only Tekla Structures for both 3D-modelling and structural analysis. Due to unknown reasons, the analysis model with information about the loads disappeared from the program in the middle of the designing stage. The project was restored using the back-up file of the model, created two days earlier. However, the modelling work done during those two days was lost.

As an example of a smaller problem that appeared while using of the analysis and design tools in Tekla, is that the synchronization of physical and analysis models are insufficient. So, despite the fact the models are in the same program, they have to be modified separately to receive proper geometry and correct calculations consequently. The next problem is that the large loads are not considered properly in structural calculations. During the interview with Ilari Pirhonen, it was concluded that both the analysis model creation and the transferring of it to FE software (software based on finite element analysis) was quite risky and unclear. Therefore, the Tekla model was exported once to FE software to create an initial FE model, then, both models have been handled as stand-alone models. The loads have been usually added in the FE software. When built-in load generation tool of Tekla Structures is used, the loads are not necessarily distributed in the right way. After the conversion to FE software, the load data in FE software is often messy and laborious to repair. Nowadays, the structural analysis is not carried out using Tekla Structures, but rather in FE software. (Pirhonen, 2019.)

To summarize, the idea of Tekla analysis and design model is a good perspective. Tekla already has the obligatory tools for accurate model transferring. Nevertheless, basing on practical experience, it is known that the system contains several mistakes leading to information losses or data corruption. In addition, the significant updates of the features are conducted only by the owner (Trimble Solutions Corporation). As follows, the user does not have enough freedom to adjust the settings accordingly with his or her own requirements. Also, the user is quite dependent on the software producer, and should wait for the further releases with fixed mistakes (Pirhonen, 2019). However, it should be mentioned that all the experience-based facts are based on using the Tekla versions launched before January 2018. At that time, the decision to concentrate on other approaches to build up the integration between the examined programs was done. Therefore, the detailed examination of analysis and design tool from Tekla was not conducted in this research.

3.4 Integration via Rhinoceros and Grasshopper

3.4.1 Software description

The programs Rhinoceros and Grasshopper are the main software used in practical part of the thesis. Rhinoceros is a 3D modelling tool working with NURBS (Non-Uniform Rational B-Splines), which are mathematical representations of 3D-geometry (Robert McNeel & Associates, 2018). Grasshopper is a graphical algorithm editor, which is add-module for Rhinoceros (Robert McNeel & Associates, 2018). In the Grasshopper the algorithms are presented as parameters modified by components (see next paragraph). The data flow is from the left to the right. Hence, the inputs are on the left and outputs are on the right. The algorithms are also called as definitions.

The Grasshopper objects are the parameters and components. The parameters store information as geometry (lines, curves, surfaces and so on), numbers, colours etc. The components are the elements containing algorithms to modify the input parameters. In Figure *7*, the interfaces of Rhinoceros, and Grasshopper are presented consequently. The example model was created basing on the curved surface drawn in Rhinoceros,

which was assigned to Grasshopper by parameter "surface". Truss elements as chords and diagonals were created by the algorithm (on the right in Figure 7). The variables are the numbers of diagonals in y- and x-directions, and the position of the intersection point of the diagonals on the lower chord. At the end, the models were created in Tekla and RFEM. The models received after transferring by components to Tekla or to RFEM are shown in Figure 8.
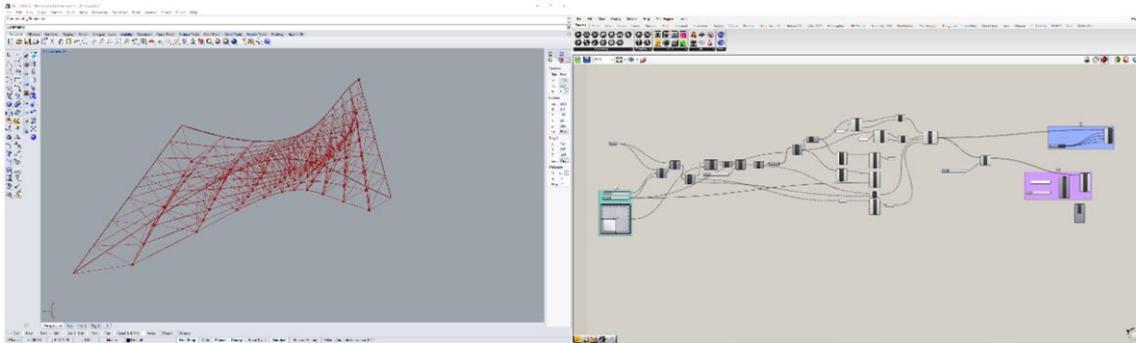

Figure 7 On the left - Rhinoceros interface, on the right – Grasshopper interface
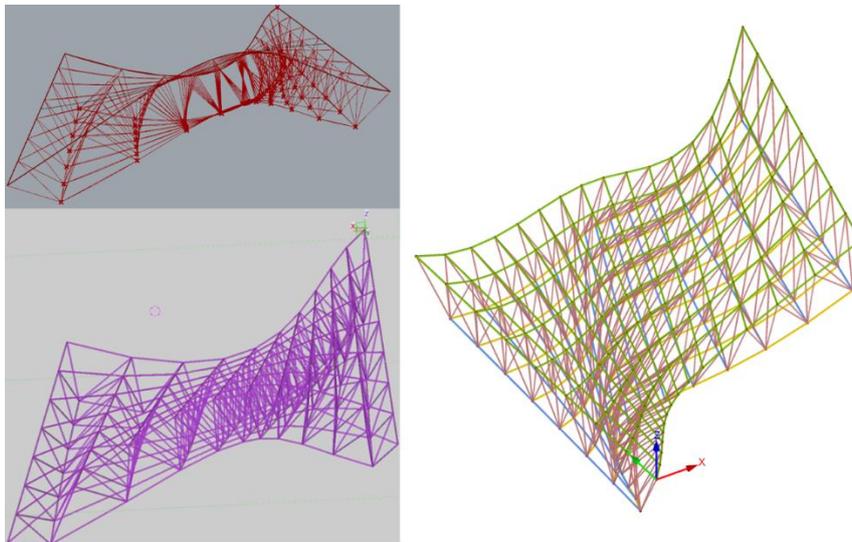
Figure 8 The geometry from Rhinoceros is in the left up corner, the models in Tekla and RFEM are on the left down corner and on the right side consequently

In Grasshopper, multiple parameters collected into one object are sorted as a list. When the data quantity is large, or when the only certain parameter from the list should be picked, the tree structure can be applied to operate the items more effectively. In Figure 9 is presented a sketch of the tree structure. The input parameters are grouped into "branches", inside which the new sub-groups can be created. The path, which describes the location of the item by identical numbers of groups to which the item belong, is assigned to each element. For instance, path number {0;1} means that the item is located in the branch with the index 0, and further, in the sub-branch with the index 1. If the number of branch levels is increased, the number of indexes is increasing as well. So if each sub-branch of the example tree is divided into smaller sub-branches, the path number is "{0;1;x}", where "x" is number of the new group in which the element is allocated. The simplest components to manage the tree structure are "Graft" and

"Flatten". The first is to create a separate branch for each item in the list. The second is to remove all branches and to structure all items as a plain list.
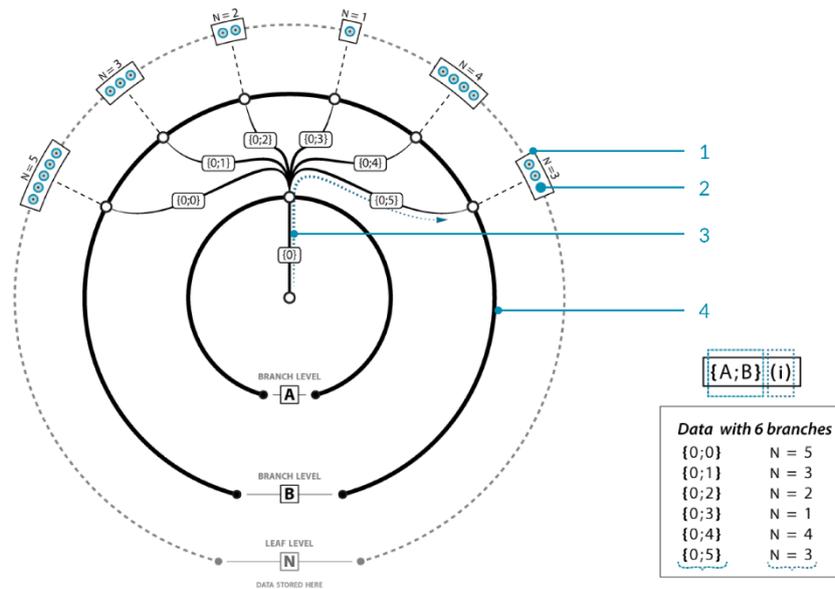

Figure 9 Basics of data tree structure in Grasshopper. 1 – list container, 2 – data items "leaves", 3 - path, 4 – current "branch" level (Mode Lab, 2018, p. 122)

Another useful feature implemented to accomplish the practical part of the research is clusters which are the components created by users. The clusters have the same structure as usual Grasshopper objects. In Grasshopper, both clusters and objects are stored in the toolbar in the upper side of the program (see Figure 7, the interface on the right). Overall, it is just an algorithm or a part of it. The option is useful, because the most frequently repeated definitions pieces can be quickly obtained from the toolbar, and besides, the main algorithm becomes less condensed.

Certainly, there are more features and tools in Rhinoceros and Grasshopper. However, the examples explained above are the most used in the new tools. The positive point is that the programs are still under development by producer McNeel Europe and by users. Therefore, new functions are included constantly, and the bugs are fixed fast. On a big scale, the software is a worthy field for settlement of the integration system between RFEM and Tekla Structures.

3.4.2   Reasons to try integration via Rhinoceros and Grasshopper

The third possibility considered during this research is connecting of Tekla and RFEM via programs Rhinoceros and Grasshopper. With this method, it is possible to avoid the problems of two other methods described above. This is because the connection is conducted by the algorithm within which all necessary modifications can be applied. What is important, the model transferring is entirely controlled by the user. That means that the engineer working in that program is not anymore so much dependent on the software producer, and can launch a variety of specific integration systems due to particular requirements. Predominantly in this study, Grasshopper tools were used to make Tekla model readable in RFEM without any manual modifications after model transferring.

In fact, Rhinoceros and Grasshopper also have components to operate the models not only from Tekla and RFEM, but also, for instance, from Revit, Karamba and ArchiCAD. Furthermore, the Rhinoceros and Grasshopper are applied in a wide range of fields, for example, jewellery modelling, structural design, software creation and so on. Consequently, the number of users is increasing fast. The software has a web-page where users can share with tools done in one graphical algorithm editor (McNeel Europe, 2019). Based on users own experience, the tools launched by the users are quite helpful.

Doubtless, the valuable extension is that Tekla Structures has special Grasshopper components to operate with Tekla elements with the help of algorithms (see Figure 10). The application of those components allows to exchange elements and data between the programs. Moreover, a new alternative design process is developed. Firstly, structural geometry made of simple curves in Grasshopper is created. Then, the variable values for some of the elements are assigned. Finally, the geometry is transferred to Tekla with the help of components, where the attributes (profiles, materials, classes etc.) are defined. After this, the geometry can be easily changed by the variables in Grasshopper. Then, the Tekla model is automatically updated. The common example of such as system is a truss structure. The geometry is represented by central lines and the variables can be height and width of the truss or the number of diagonals.
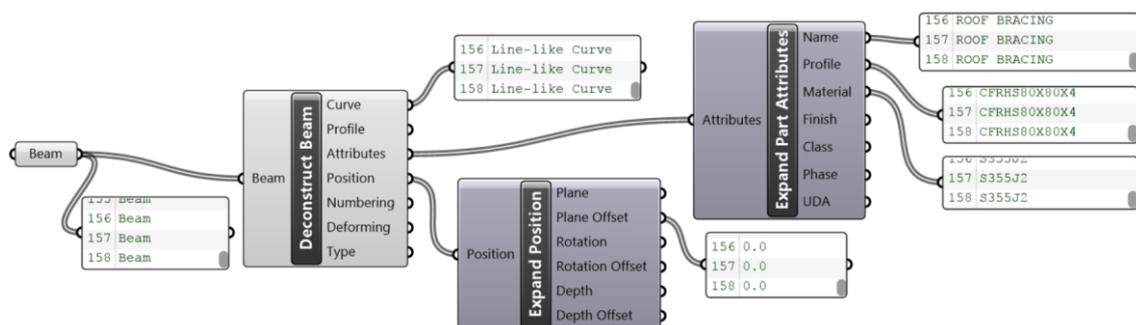

Figure 10 Tekla components made for Grasshopper

By this time, there is no official module of RFEM components in Grasshopper. Instead, such tools are developed by structural designers. The components used in this study were created by the company AINS Group. They were used in the projects completed by the company, when the parametric design was involved. For instance, Kansi Areena in Tampere was designed involving algorithms in Grasshopper (A-Insinöörit, 2018). The logic and methods of the RFEM components are similar to those from Tekla. Initially, the components were repeating the main functions of RFEM. Therefore, while transferring model with the help of Grasshopper, the same problems as after the direct data exchange (see Chapter 3.2 "Direct data exchange between Tekla Structures and RFEM") appeared. In contrast, those mistakes could be corrected using algorithms with the minimum of manual model editions.

In conclusion, the approach of creation and development integrated system in Rhinoceros and Grasshopper was chosen as the object of this research. It was obvious that it was not possible to set all the system during the time given for the study.

Therefore, it was decided to create several tools solving the most common of the problems with model transferring via RFEM exchange system. The descriptions of those tools can be found in Chapter 4.3.2 "The logic of the algorithm", and the detailed descriptions of the algorithms are in appendices.

3.4.3   Types of integration via Rhinoceros and Grasshopper

To sum up previous information about the topic, two main programs (Tekla Structures and RFEM) with two models of the same designed structure are considered. The models should be the same in order to get correct project data. The modifications done in one of the programs should be somehow transferred to the other. The Rhinoceros program with add-on called "Grasshopper" is implemented to set the integration between the programs. The implementation level is chosen for each project separately due to demands and specifications. In case of connection between models from RFEM and Tekla Structures, the practical project cases have shown different levels of integration. Here are presented three integration levels that have been applied in some real projects (Pirhonen, 2019). The concept ideas of the different integration levels with the help of algorithms are here called as "full integration", "heavy integration" and "light integration". The advantages and disadvantages of the listed types are described in the next paragraphs.

The "light integration" level contains the simplest algorithm from the considered levels. There are two separately modelled structures in Tekla and in RFEM. The only connection between the models is presented by checking systems situated in Grasshopper. The idea is to set profiles, materials, coordinates and other attributes chosen depending on requirements. The algorithm is aimed to check that the listed characteristics of the correlated elements in two models would be the same. The main purpose of the algorithm is the reduction of probable conflicts between BIM model and FEM model. The application of the tool can be done right away; hence, the models do not require any special preparation. That leads to the availability of completing the first check even at the end project phase. The modifications are carried out manually in both models.

The second level is "heavy integration". In this case, the initial geometry is modelled manually, but only in one program (Tekla Structures). The analysis model is formed using an algorithm, which uses Tekla model as an input. Firstly, the elements from Tekla are brought to Grasshopper. The received model is "cleaned", i.e. the geometry is modified to be suitable for analysis and the unnecessary information is filtered. After the necessary modification, the model is transferred to RFEM. From the RFEM, after the analysis done, profiles and materials are modified in Tekla Structures according to the received results. This type is taking more time to complete the algorithms than the first one. On the other hand, RFEM model is updated automatically, if anyhow the initial Tekla model is changed.

The "full integration" is totally automated connection of two models. All the changes are properly transferred to all the models without information losses with the minimum level of human help. This is the ideal system. However, at the current achieved stage of integrated design, it is taken a significant amount of time to set up such a system. On

the other hand, once it is completed for the project, it can usually be applied with few modifications in other similar projects. This type of integration fits the best to a project which is algorithm controlled, starting from the geometric determinations. In addition, there are not much complicated details taking plenty programming time in the model. Another reason is expectation that the several structures can be produced using the algorithm with only minor modifications. Moreover, the developed algorithms (or their parts) might be turned into new components which can be copied to other projects. Therefore, despite larger time losses on development of data exchange in one project, the appeared tools pay themselves back in time saving in the future.

4   DEVELOPED TOOLS

In the previous chapters the problems of data exchange methods between RFEM and Tekla Structures made by Dlubal Engineering Software (see Chapter 3.2 "Direct data exchange between Tekla Structures and RFEM") and Trimble (see Chapter 3.3 "Analysis model in Tekla Structures") are described. The connection with the help of Rhinoceros and Grasshopper also have a lack of components to complete transferring accurately enough. Nevertheless, the algorithmic design has enough opportunities to build up a properly functioning integration system. Therefore, the practical investigation of the exchange method via Rhinoceros and Grasshopper was completed as a part of this research. Additionally, several new tools to reduce the amount of manual modifications were developed. Below, in this chapter, a detailed description of practical part can be found.

4.1   Effective length

4.1.1   Buckling resistance

The buckling is the deflection which occurs due to compression. For example, columns are the structural elements subjected to axial compression, so their buckling resistance should be checked. The critical load is the maximum axial load that a column can support when it is on the verge of buckling. The Euler´s formula (see Formula 1) is used to determine the critical load of the ideal column (see Figure 11). "An ideal column is initially perfectly straight, made of homogeneous material, and the load is applied through the centroid of its cross section" (Hibbeler, 2010, p. 690).
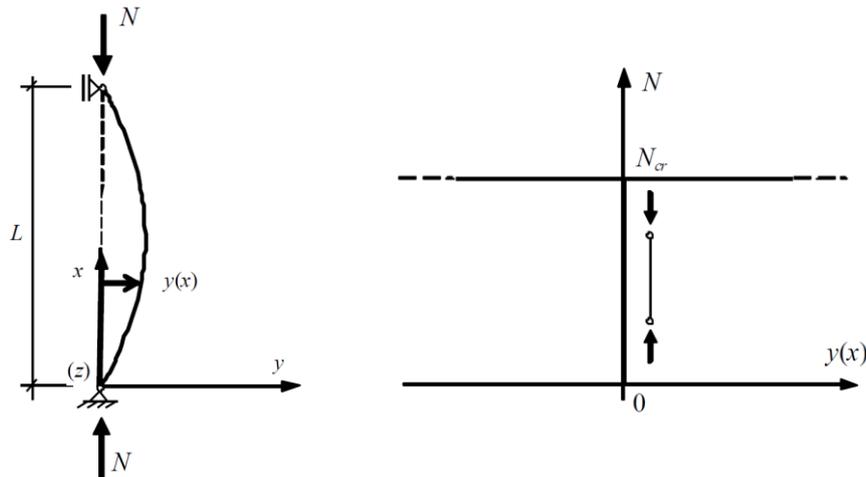
$$P_{cr} = \frac{\pi^2 EI}{(KL)^2}$$
(1)

Figure 11 Buckling in a pinned member (Euler´s column) (Simões da Silva, Simões, & Gervásio, 2010, p. 173)

The buckling length or the effective length is system length of an otherwise similar member with pinned ends, which has the same buckling resistance as a given member or segment of member (Finnish Standards Association SFS, 2005). Usually, determining of the effective length is done manually. Because for example, the effective length of the segments in the column connected to the bracings in the middle will be equal to the full length of the column in X-direction or to the half of that in Y-direction (Figure 12). Because the neutral axes of the bracings are situated along XZ-plane, they influence the moment about Y-direction.
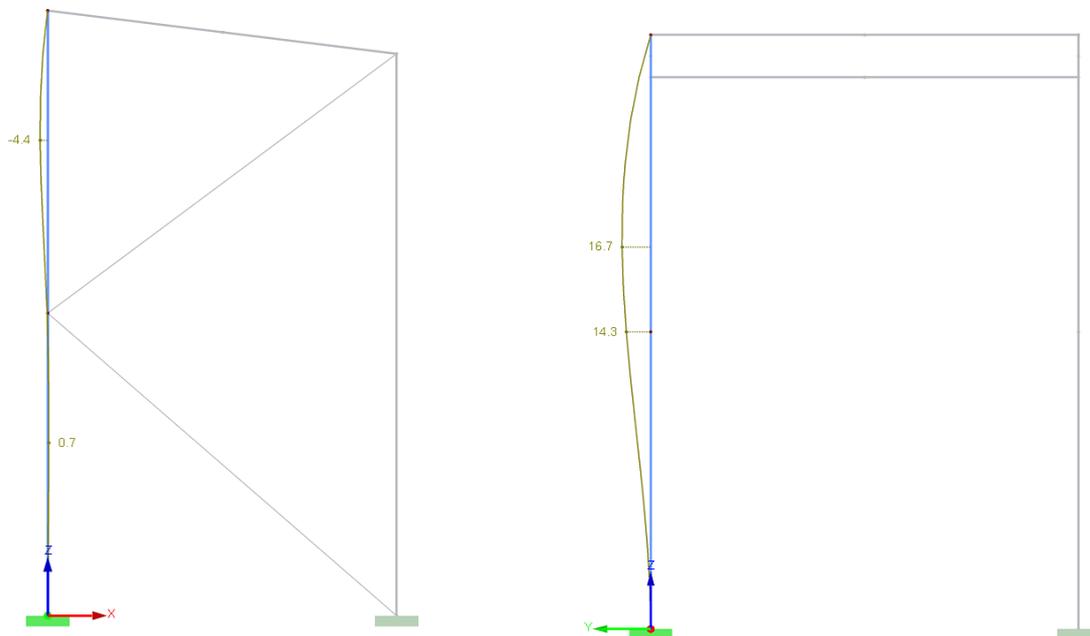
Figure 12 Buckling of the column in X- and Y- directions

In the RFEM, it is possible to set the effective lengths or coefficients of effective lengths for each member separately. The buckling lengths are taken into consideration in add-on modules like RF-/STEEL EC3 where stability analyzes are performed (Dlubal Software GmbH, 2017). Generally, elements like columns, beams or trusses will be represented by members obtained from the initial elements by dividing them into segments in places

of connections, intersections or supports. The default buckling lengths of each segment in both directions will be equal to the full length of the segment. The correct buckling lengths or factors should be set manually using additional settings "Member Eccentricity" located in "Edit Member" window, "Options" tab (see Figure 13)
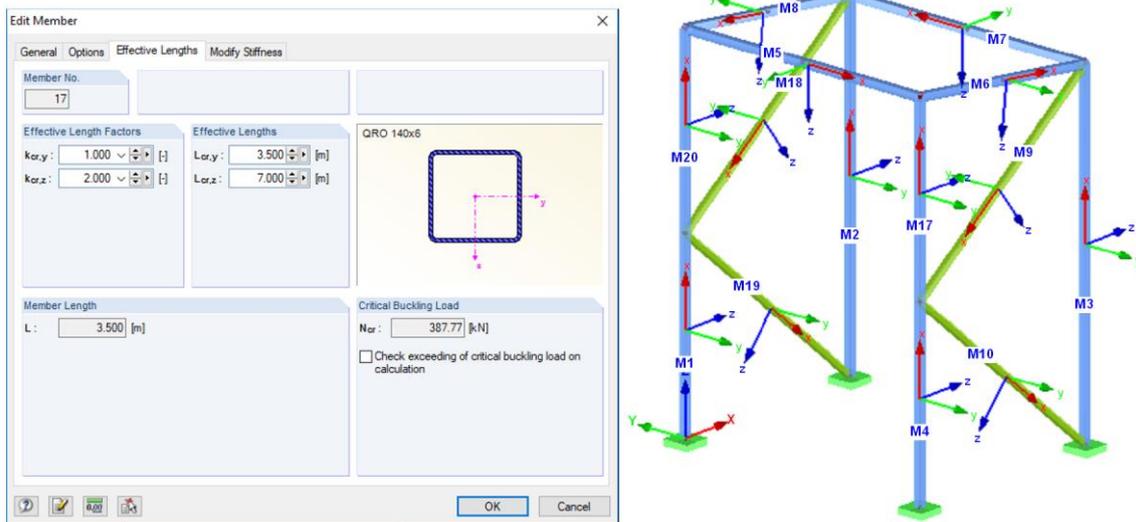

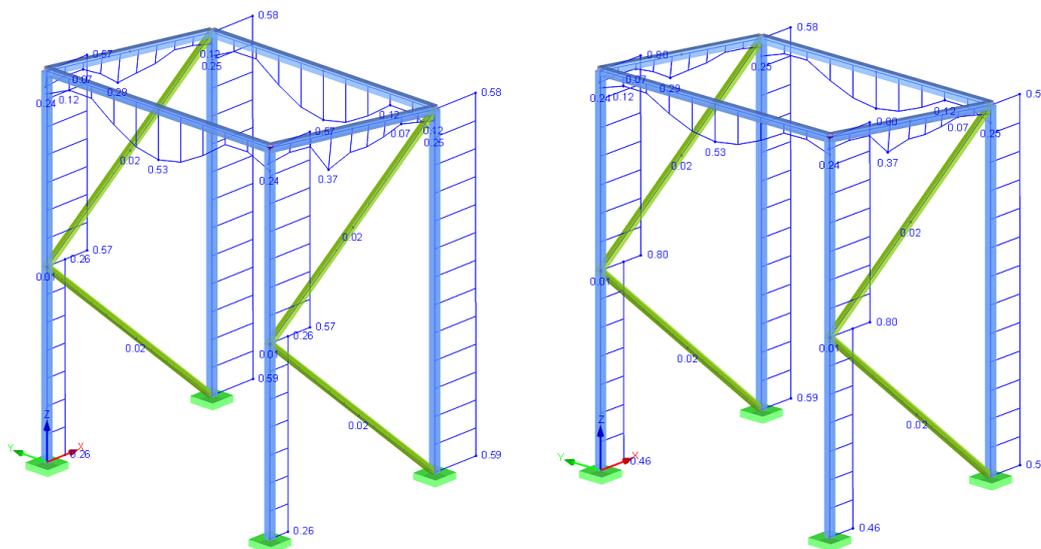Figure 13 Settings of the effective lengths (member M1)

Figure 14 The result from RF-STEEL EC3 add-on module. On the left - with default effective lengths, on the right - with modified effective lengths

In Figure 14, the results of RF-STEEL EC3 add-on module are presented. The buckling lengths of the columns restrained by bracings in the middle were corrected in the right model. Therefore, the design ratios, defined by dark blue numbers and diagrams, are higher for those columns. Because in local Z-direction, the effective lengths are twice increased, therefore, the stronger cross sections are demanded. The smaller the buckling length, the weaker element (smaller cross-section) can be used. That means that if the lengths of the segments are used as effective lengths, the structural elements are designed weaker than they should be after the buckling resistance check. Contrary,

if the full lengths of the elements are used as the effective lengths for each segment into which they were divided, the structural elements will be designed with over dimensioning.

The problem is that there are usually plenty of members where such value should be modified. The process is not automatic by now. Therefore, it is one of the time-consuming parts of the designing process. Moreover, if some changes were done in the structure so the lengths were modified, the effective lengths corrections should be completed again. The proper checking should also be done after every modification to avoid mistakes since the process is manual.

4.1.2 "Effective length" component

Considering all above, the tool determining the correct effective lengths of the columns is created. The model geometry and the effective lengths are modified using Grasshopper. The component "L.eff" (Figure 15) contains the algorithm which divides the columns into segments to work with RFEM and determines the effective lengths for each new member (Figure 16).
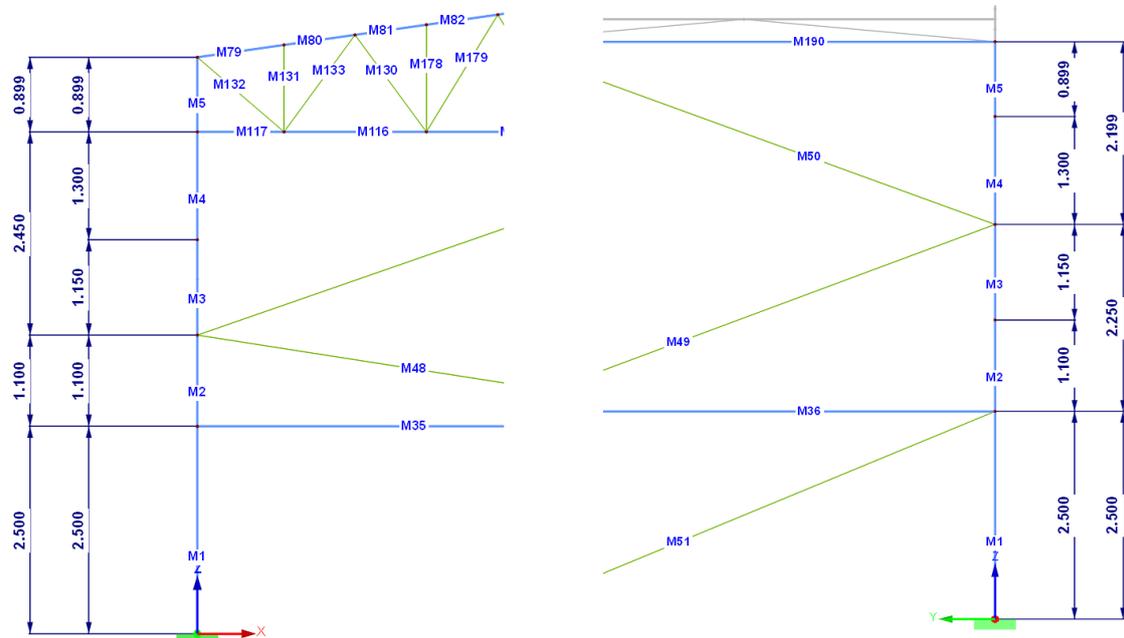

Figure 15 "Effective length" component

Figure 16 Dimensions of the effective lengths in global Y-direction (left model) and global X-direction (right model)

The inputs are named as "CrvCOLUMN" and "Crv (DIVing)". The first is the set of column-elements, which is modified by the definition. The second is a set of restraining elements. Both inputs should be represented as curve- or line-elements. The outputs "C(CL_DIV)" is set of segments for RFEM created by splitting columns at the

intersections. The "L.eff_Y" and "L.eff_Z" are sets of separate effective lengths in local Y- and local Z-directions correspondingly. All three outputs have the same number of items. The items with the same indexes are related to the same segment.

To use the component, the elements from Tekla model should be referenced into Grasshopper via "Beam" Tekla component. Then the column, beam, bracing and chord-elements are picked from all elements applied in the Tekla model. Then, the elements are exploded by the other Tekla components in order to obtain sets of curves, which are the central lines in Tekla. To use the received data from the "Effective length" component, the outputs are connected with the RFEM components as inputs.

4.2 **Offset settings**

4.2.1 Eccentricities in the joints

In case of connections between different members of a structure, if the force applied does not pass through the center of gravity of the joint, then such joint carries moment in addition to an axial direct force. Such types of connections are called as eccentric connections and the eccentricity is calculated as the bending moment divided by the axial force (Figure 17). From the opposite side, the bending moment is axial force times the eccentricity (Millais, 2017.) Most constructions tend to use flexural members functioning in bending, as the dominant structural elements (Wyatt & Hough, 2013). Therefore, the eccentricities should be defined properly to receive the correct estimation of the bending moments. Obviously, the bending is taken into account in RFEM calculations, which are based on classical mechanics considerations and requirements of the structural standards.
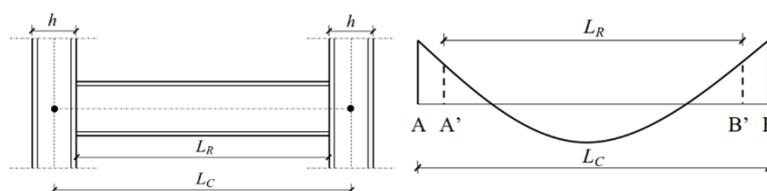

Figure 17 Influence of eccentricities (Simões da Silva, Simões, & Gervásio, 2010, p. 38)

In this study, the eccentricities in the connections were considered due to the shape of the elements. The examples of the connections are in Figure 17. Sections AA` and BB` are rigid, therefore, the maximum negative moments of the elements should be evaluated at points A` and B`. If parts AA` and BB` are assumed not to be rigid, it can be seen from the moment diagram, the moments at intersection points of the central axes A and B are bigger than at points A` and B`. Thus, neglecting the elements shapes leads to over-estimation. "For a uniformly distributed load, the difference reaches 19% when h/LC = 0.1(that is, for a 6 m span between axes and IPE 600 columns, for example)" (Simões da Silva, Simões, & Gervásio, 2010, p. 38.) In addition, the same problem due to the physical shapes appears in the connections shown in Figure *18*.
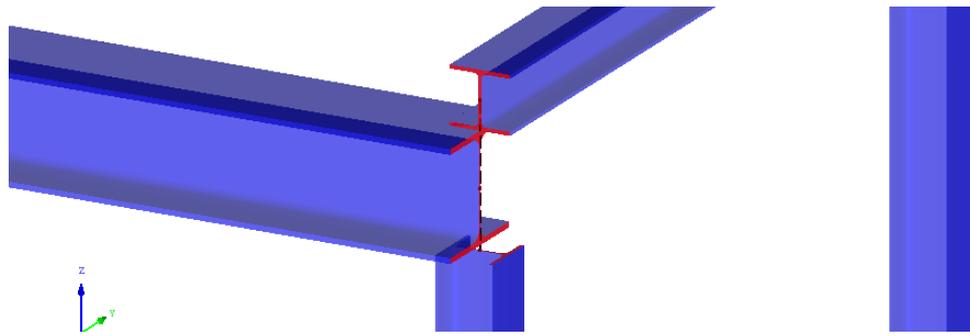
Figure 18 Example of connection

In the RFEM, the eccentricities can be identified as rigid links. A rigid link is created by "rigid" member type which couples displacements of two nodes by means of a rigid connection (Dlubal Software GmbH, 2016). To create the rigid link, the new member should be done. Then, the member type should be set as "Rigid". Another way to define eccentricities is to use additional setting "Member Eccentricity" located in "Edit Member" window, "Options" tab (Figure *19*).
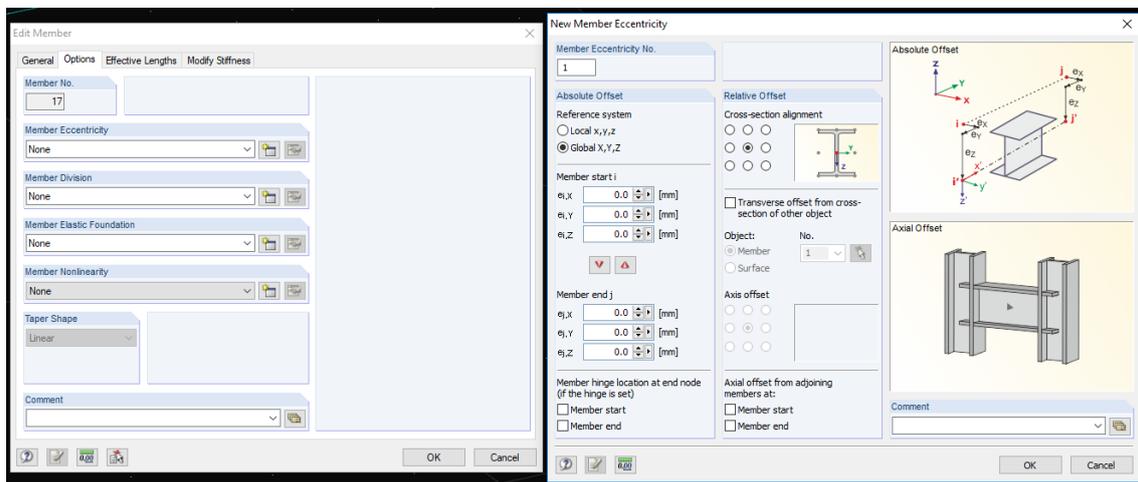

Figure 19 Member Eccentricity in RFEM settings

When modelling in Tekla, it is possible to change the location of the element by modifying the offset distances from the neutral axes. Initially, the model brought from Tekla to RFEM via direct link or to Grasshopper via components keeps only the neutral axes without offsets. Consequently, there are significant differences in geometry between models in the used programs. Therefore, the received model should be modified manually to make it similar to the one in Tekla. Another option is not to use the offsets from neutral axes in Tekla model placing all the elements directly into correct places i.e. that will be an additional restriction to the modelling. In addition, eccentricities in the trusses or in structures as presented above are easy to set with the offset settings. Then, rigid links should be added to the RFEM model between the end points of the neutral axes and of the elements. The process is also manual. In both cases, the human-work is demanded at several phases, which causes additional designing time for editing and checking. Consequently, the tool turning such modifications into an automatic process increases speed of the process and reduces the number of human errors.

4.2.2 Description of the algorithm

Because of the disadvantages of the current system related to the offset setting transferring, the Grasshopper-component was created (Figure 20). The first input is the Tekla beam-component, which is the set of Tekla members with modified offsets. The second input is a set of columns, which are not modified by this algorithm. The column elements are demanded to delete the rigid members in connection within the column, but not at the ends of it. The outputs are the members with correct position settings for RFEM and the rigid links in correct places. Then, when a model is loaded to the RFEM, the structure has the same initial geometry with rigid links in places of the offsets which affect further analysis.
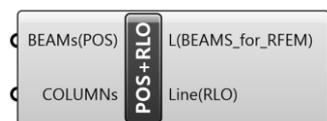

Figure 20 "Rigid links and offsets" component

To present the modifications described in previous paragraphs, the structure shown in Figure 21 was used as an example. The algorithm recognizes the type of displacements, changes the member position and creates the rigid links while transferring to the RFEM. If model in TEKLA is modified, for example, the profiles are changed, the structure is automatically updated in the RFEM.
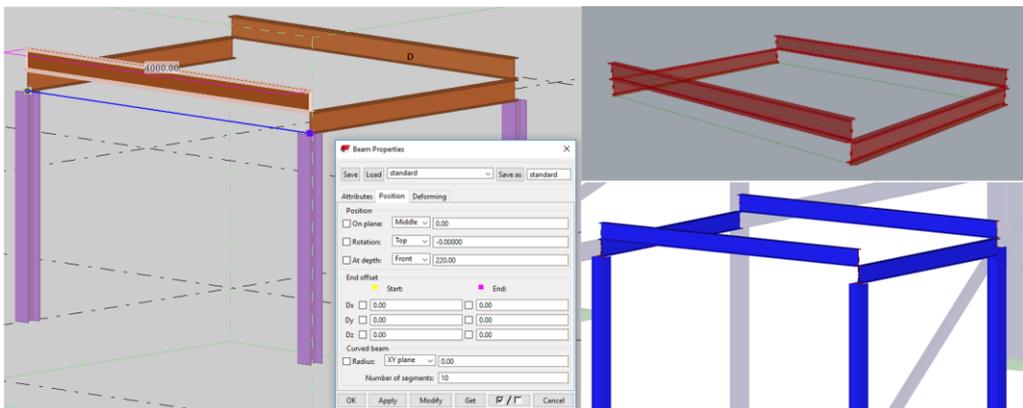

Figure 21 Example of the connection. The models are from Tekla (on the left side), Rhinoceros (in the right upper corner) and RFEM (in the right lower corner)

4.3 Profile and Material converter

4.3.1 Data mapping between libraries of Tekla and RFEM

Tekla Structures and RFEM are developed by different companies. One consequence is that there are different libraries of profiles and materials in the programs. If the parameters of an element, cross-section or material, are not named the same in both programs, the properties related to these parameters cannot be transferred from one program to the other; hence, these elements cannot be involved in further calculations.
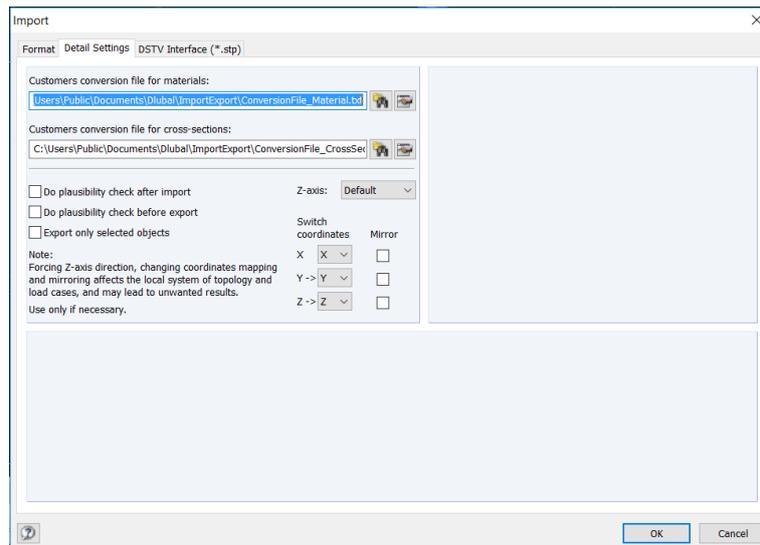
Figure 22 Conversion files

One of the ways to solve the problem is to set the pre-installed by RFEM conversion files for the materials and profiles (see Figure *22*). The process of setting is just writing in one line the name from TEKLA, and then, the correlated name from RFEM (Dlubal Engineering Software, 2010). The method is slow, because names for each profile type (see Figure *23*) and size and for each material should be added manually.

Figure 23 Example of input for the profiles

The alternative method is to use parametric design. The component for data mapping between the libraries (profiles and materials) of the programs is shown in Figure 24. The input is the set of beam-elements from TEKLA. The outputs are names of cross-sections (CrSc) and materials (Mat) for RFEM. In Figure 25, filled data about the profile and material, which were both changed in Grasshopper with the profile and material converter, are presented to prove that the names are assigned properly.
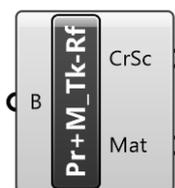

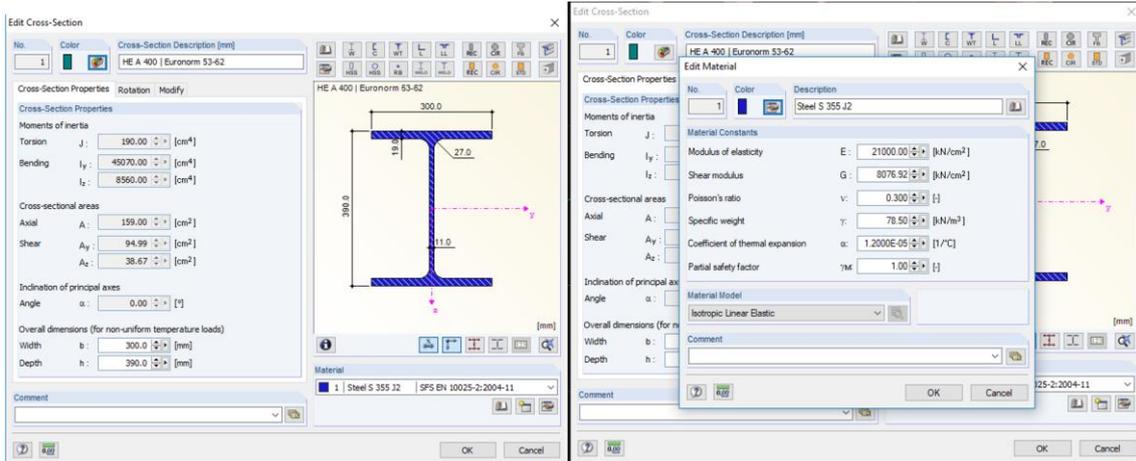Figure 24 "Profile and Material converter" component

Figure 25 The automatically filled data about profile (on the left) and material (on the right)

4.3.2 The logic of algorithm

In general, the target is to identify correlated names of the same profile or material from Tekla Structures and RFEM. Then, in case of transferring model from Tekla to RFEM, the names should be modified to fit the correct names in the library of RFEM. The indexes were implemented to set such mapping (see Figure 26). The indexes are represented by path numbers of the material or profile names.

Figure 26 The lists of Tekla profiles, indexes at first stage, corrected indexes and RFEM profiles (correspondingly from left to right)

As follows, the process of name changing is based on picking the correct replacing text with the same index as the index belonging to the replaced text. Then, after correlation was defined, the text is edited. The component "Match Text" is used to find the demanded values. Preliminary numbers are based on the differences in Tekla names. However, the additional challenge is that for some of the profiles the names are the same in Tekla, but different in RFEM. For example, the steel rectangular hollow sections are named "CFRHS" in Tekla, and "RRO" or "CHS" for rectangular and square profiles consequently in RFEM. The indexes are additionally modified in such cases based on the fact that the Tekla names of the rectangular profiles contain two values for the width and height of the cross-section, while the names of the square profiles contain only one value due to the equality of the dimensions. In the same way, L-profiles are different.

After the names are set correctly, the received lists are referenced to the RFEM components. The algorithm works with a multiple data inputs so that input path

numbers of items remain unchanged. Therefore, the updated names are assigned exactly to the same elements without any corrections.

4.4 Additional tools and systems

4.4.1 "SNAP" components

Regardless of how the model is transferred to RFEM from Tekla, there are often several wrong nodes appearing due to inexact intersections of the central lines (see Figure 27). If Grasshopper is applied as a link between Tekla and RFEM, it is possible to set the tolerances for the connections. So, some maximum limits for these offsets, when the deviations will be neglected, are defined. Thus, the lines, which are almost intersecting in Tekla, have an intersection node in RFEM. However, if there are more than two members almost intersecting at the same place, they are not connected by one node with the help of tolerances. Another decision is manual modifying of the model in RFEM, which is hardly applicable due to a frequent updates of the whole model.
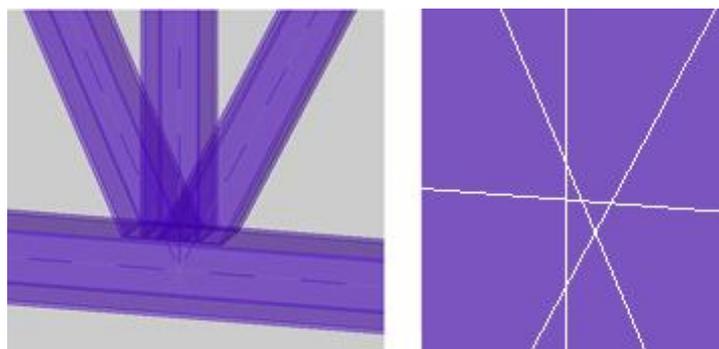

Figure 27 Inexact interceptions in Tekla model (on the left – the connection of the truss diagonals and the lower chord, on the right – the zoomed central lines of the elements at the connection)

Figure 28 "Snap for truss" and "Snap to perpendiculars" components

To solve the problem, the additional components were developed in Grasshopper. In Figure 28, the component for the truss diagonals is on the left side, and the component for the snapping to locating in one plane members is on the right. The main purpose of the first components is to set correct positions of the diagonals in the trusses, because there are commonly connections of more than two members in such assemblies. The component for the trusses "Tr_SNAP" contains the inputs for the sets of lower chords, top chords and diagonals (labelled as CrvLC, TCh and DIAG correspondingly). All the elements in the sets are curves. The output is a set of modified diagonals, with their end points located on the chord's central axis. The tool can be used with one- or two-sloped roof structures. In the second alternative, sloped chords are divided into two groups so the elements in one group have the same slope. The second component was applied for

the roof bracings to place the end points of the bracings lying on the roof beams. The inputs are sets of roof beams "RoofB" and roof bracings "RoofBR". The output is set of modified roof bracings.

4.4.2   System of division by member-type

During the design process, there are different types of elements used (beams, columns, trusses etc). The division is done due to differences in functions and properties in the structure. For example, beams are members that support perpendicular loads in relation to their axes, while columns are long slender members that support axial compressive loads. (Hibbeler, 2010.) It should be noted that such division is quite general. For example, columns can carry horizontal loads such as wind load.
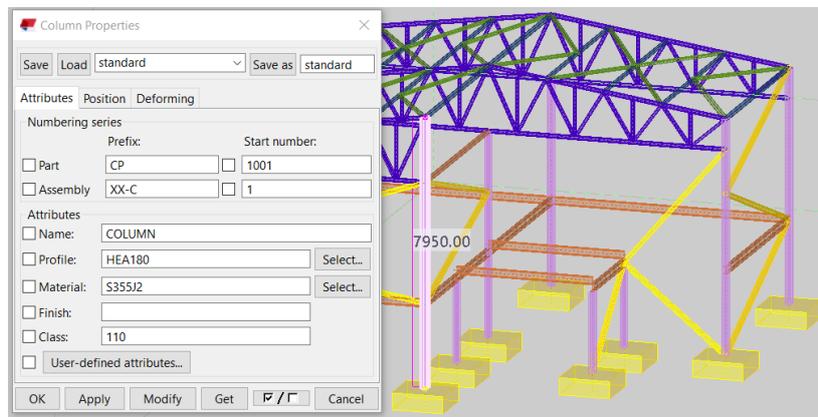

Figure 29 Column Properties in Tekla

In Tekla Structures, the types of elements can be defined manually by adding the correct name in the "NAME" field from the element's properties window (see Figure 29). In RFEM, the member types should be defined accordingly with the forces which they are transfer. For instance, the type "Beam" defines a rigid member which is able to transfer all internal forces, the type "Truss" is a beam with hinges located at the member ends which do not transfer any moments (Dlubal Software GmbH, 2016). Each type has a different influence on further calculations. Therefore, in order to obtain correct results, the correct member types should be chosen from the "Edit Member" tab.
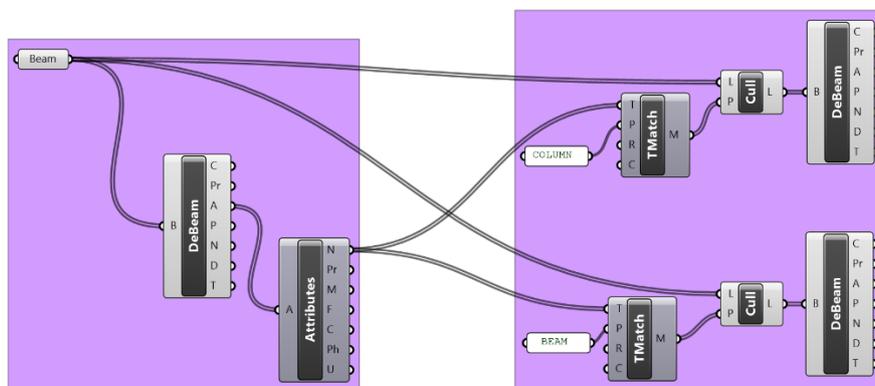

Figure 30 Elements division by names

To avoid additional modifications, the division of the elements was set to be automatic in Grasshopper (see Figure 30). To make a separation while transferring, the correct names should be assigned to all the elements in TEKLA. Then, the elements are brought to Grasshopper using "beam" Tekla component. Therefore, all the structural elements are presented as lines with assigned information of the Tekla element. That means that the geometry is presented by the neutral axes of the elements (see Figure 31). The information including assigned names of each element can be obtained by using the "Deconstruct Beam" and "Expand Part Attributes" components (labelled in the Figure 30 as DeBeam and Attributes correspondingly). Then, the elements are divided into groups by names. After this, the necessary geometry modifications are made. At the end, to set the correct member types for RFEM, the groups of elements are connected separately to the RFEM components, and to each group the correct member types are assigned. As a result, the model in RFEM has correct settings for the member types right after transferring.
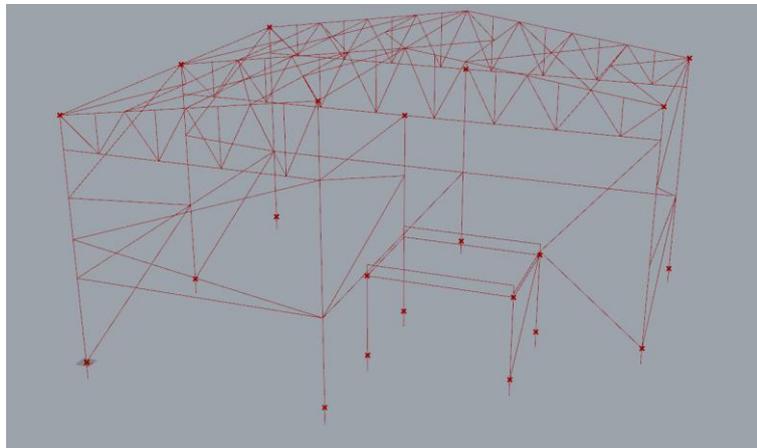

Figure 31 The example Tekla model transferred to Rhinoceros

4.4.3   Match checking

Originally, the concept of checking the geometry in the models from Tekla Structures and RFEM was developed within the "light integration" type (see Chapter 3.4.3). However, due to the critical importance of the similarity between models, such a tool should become a part of all integration levels. The principal idea is to retrieve the elements from Tekla and RFEM into Grasshopper. Then, the sets are compared in order to find the dissimilar elements. The values to be compared are coordinates and, maybe, GUIDs (Globally Unique Identifiers). The GUIDs are presented as random numbers assigned to each element. They are involved in the checking, if they are somehow correlating in both two models. The example case is when the initial model is Tekla model, which was transferred to RFEM. In this case, the GUIDs assigned in Tekla are exported with the model to RFEM. Another option is to define the GUIDs manually.

Firstly, the case when coordinates and GUIDs are checked is described. The process starts from referencing the Tekla model to Grasshopper. Then, the GUIDs are defined for the elements with the help of "GUID assigning" component developed by AINS Group. In Figure 32, the possible parts of the unique numbers are presented. In addition

to random numbers, the GUIDs can consist of the assembly numbers, position numbers, names and all other basic attributes of the Tekla elements. Thus, after the import to RFEM, the created members have assigned Tekla attributes, which simplifies the correlation between two models. For example, the analysis results from RFEM can be directly identified with the elements in the Tekla drawings with marked assembly and position numbers. Further modifications of the models are done separately. However, the match check is carried in the Grasshopper. The algorithm matches the elements with the same GUIDs, and checks that those elements have the same coordinates.
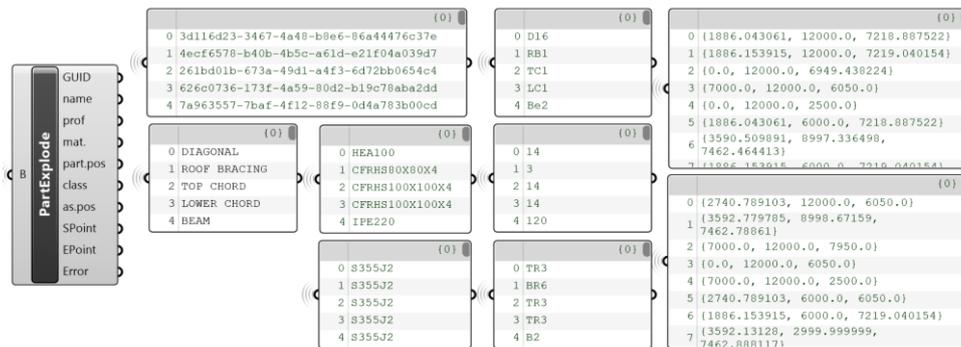

Figure 32 Component assigning GUIDs

Contrary, the case when only the coordinates can be considered, has challenges which put under question the possibility of reliable results obtained from the checking. The biggest challenge is the absence of any characteristic which would always have equal values for the same elements in Tekla and RFEM. In fact, even the coordinates of the same elements placed correlatedly in two models vary sometimes due to the inexact connection points in Tekla software (see Figure 27). One of the decisions is to apply tolerances, thus the negligible deviations due to software are not taken into account. The next thing is that the physical and analytical models have different logic in member division. So, a single column from Tekla Structures should be split at points of intersection with other elements and only then, the obtained segments are sent to RFEM. Therefore, the numbers of members in two programs vary. The problems listed above and the match check in such cases are the aspects which are planned to be developed in the future.

4.5 Steel structure – case study

The aim of this chapter is to show how the developed tools can be applied for one model. The example model is an imaginary small building with the superstructure made of steel. Initially, the structure was created in Tekla. The correct names were given to the elements, the different profiles and steel grades were applied, and the offset setting were used for several members. Then, the model was transferred via Grasshopper algorithms to RFEM. According to the definitions of the integration types (see Chapter 3.4.3 "Types of integration via Rhinoceros and Grasshopper"), the "heavy integration" was applied for this model.

Firstly, the model was transferred to RFEM via direct link (see Figure 5). The common problems (see Chapter 3.2 "Direct data exchange between Tekla Structures and RFEM")

appeared: wrong profiles and materials, incorrect positions of the members with set offsets, absent division of elements into segments, lost member types etc. Consequently, the analysis of the model is not possible without fixing these problems. Moreover, the additional setting for the buckling lengths and offsets in the connections should be completed. The case has demonstrated the main disadvantages of the direct link transfer.
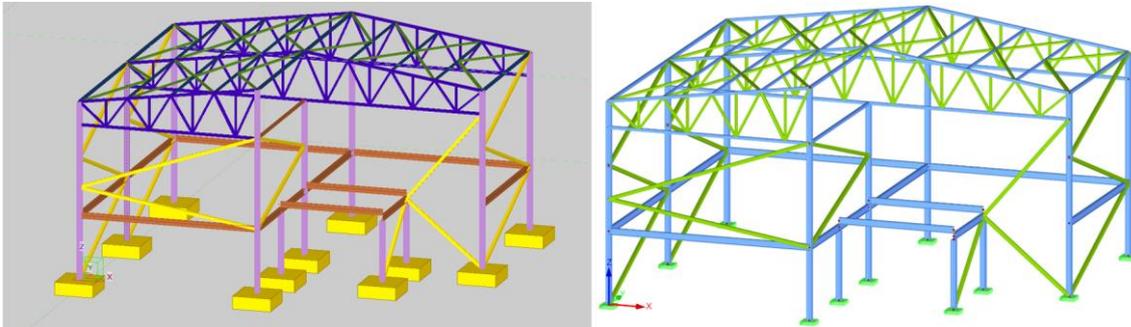

Figure 33 On the left - the Tekla model created manually, on the right - the RFEM model created with Grasshopper application

The second phase is transferring the same Tekla-model to RFEM via Grasshopper applying new tools with the already existing ones (see Figure 33). The received RFEM-model has correct geometry, the profiles are located at the correct position, taking into account the offsets of the neutral axes due to the size of the cross sections (see Figure 34 and Figure 33), and that all the elements are divided into segments at the intersection points. The connections of more than two members have one node per each. The member types are assigned according to their names. In the right model (see Figure 33), the member types are shown in different colors. The blue, green and white colors are "Beam", "Truss" and "Rigid" member types respectively.
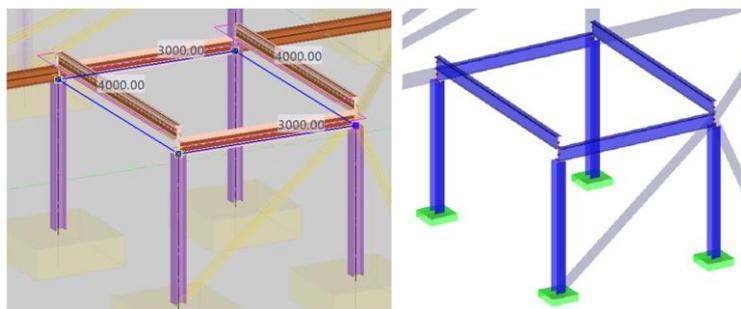

Figure 34 Elements with eccentricities

In addition, the settings of the effective lengths were done correctly for the columns. For example, the column segment, shown in Figure 35, has a total length of 1.1 meters. However, different buckling lengths were assigned separately for y- and z-directions. Some of the members had equal buckling lengths in one of the directions. The reason was that the restraining at the node connecting the element affected only the length in other direction. For other members, the values were given the same as the full length of the segments.

The next additions were the rigid links. They were created at the connections of the elements with the modified offset settings. In the example model, there were four beams with offsets in a vertical direction (see Figure 36). To simplify the work, the beams with the offsets were retrieved into a separate group from the other beams by an algorithm, which was picking the elements with the applied offset settings. The rigid links were added into the connections, which were at the column ends. Therefore, in the connection lying in the middle of the column there was no new member created. In Figure 36, the second connection has no rigid member, rather the column segment (numbered M20 in the scheme) between the intersections with the upper and lower beams.
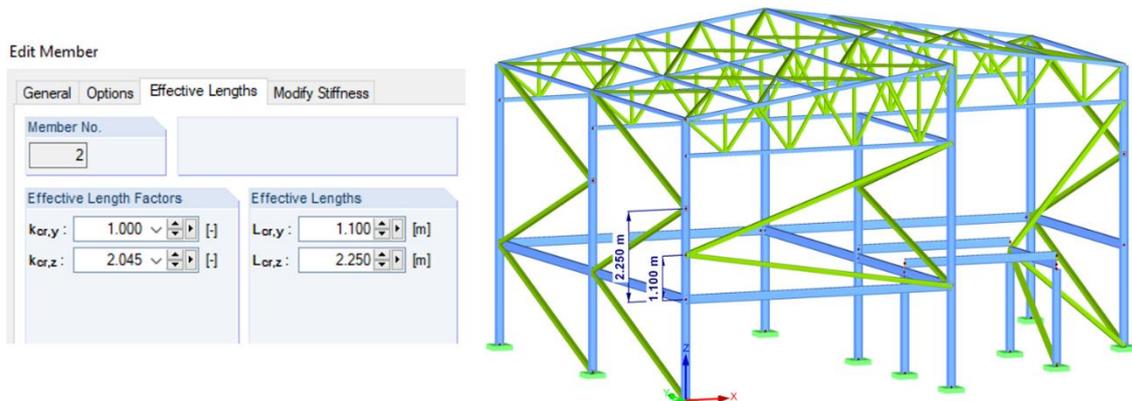

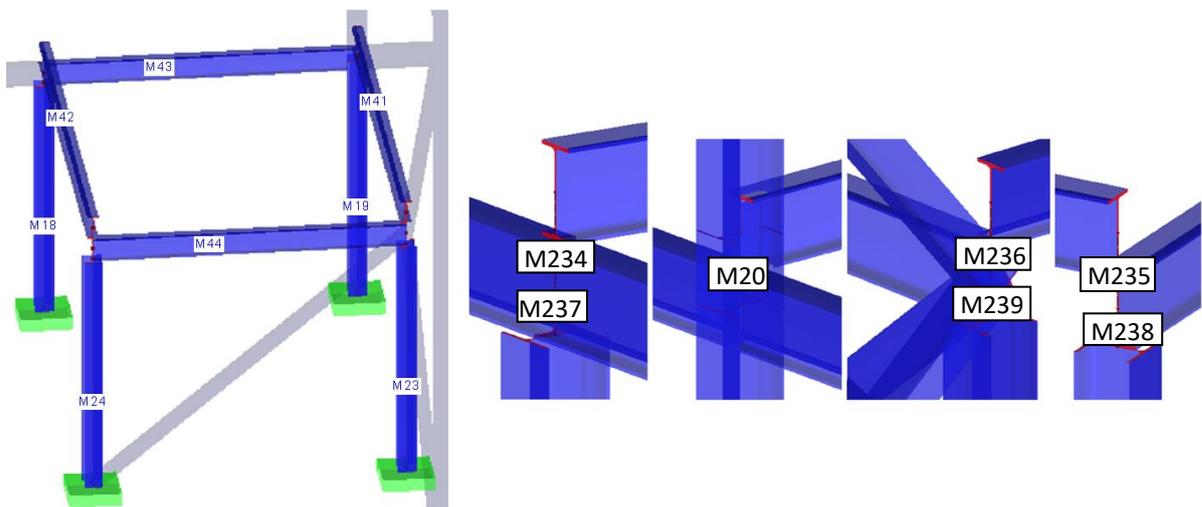Figure 35 Effective lengths of the column segment

Figure 36 Rigid members in the connections with the offsets

After applying all new tools in addition to already existing components, following the description above, the correct geometry in RFEM is obtained from Tekla via Grasshopper exporting algorithm. The received model is subjected to further analysis without modifications. Each of the created components can be implemented in other algorithms. If the initial Tekla model is changed, the RFEM model is changed automatically. Also, the whole definition can be applied with other models. Therefore, the result of the practical study is the geometry-setting system related to "light integration" level (see Chapter 3.4.3), which is flexible enough to be developed further on.

5   CONCLUSION

In conclusion, the integrated design systems are beneficial. For the purpose of structural design project, the method is based on the connection of all data from utilized software. Thus, in an ideal case, the changes done in one program directly affect the data from all other programs. As a matter of fact, the properly created integrated system reduces the time spent on the design and the amount of manual work. In addition, the part of the considered method is parametric design, which allows to set the often-edited values as the variables influencing the structure with the help of algorithms and mathematical equations. Therefore, the complex structures based on mathematically defined shapes are calculated almost as fast as the common structures with the application of integrated tools.

Due to the fact that the topic is wide, only the integration between Tekla Structures and RFEM was considered in detail. In particular, three main methods were estimated – using RFEM option "Direct data exchange", creating the model with the help of Tekla option "Analysis and Design model", and referencing model to additional software called "Rhinoceros" and "Grasshopper". In summary, the first two methods are not developed enough to implement them in real projects. RFEM data exchange tool has a lack of modifications and converters available, so the model is exported from Tekla to RFEM with significant mistakes. With the second method, the analysis model which is created directly in Tekla can be calculated in the same program or exported to RFEM. The RFEM model received from Tekla analysis model is more geometrically correct than the one received from Tekla physical model. However, several problems remained. On the contrary, the third exchange method only provides a set of tools (or components), which are located in Grasshopper. It should be mentioned that the Rhinoceros itself is almost not involved into connection settlement, since this program is for 3D modelling, while the data modifications are conducted via algorithms from Grasshopper.

In fact, one of the advantages of Rhinoceros and Grasshopper is that the programs have user-friendly interfaces. A short period of time is required (about two months) to start working with the software. For the case study the "heavy integration" type is chosen. The algorithm was set using the official Tekla components for Grasshopper and the RFEM components created by AINS Group. The first received RFEM model after the transferring from Tekla Structures via Grasshopper algorithm has similar problems as the models received after the export with the other two methods. Contrary, in case of application of Grasshopper, the problems can be solved by creating additional algorithms. Since the problems are common for most of the projects, the new components consisting of the definitions, which make data readable properly in RFEM, are stored in the same tab as the standard components. Therefore, they can be applied directly in other projects from the Grasshopper toolbar.

The connection via Rhinoceros and Grasshopper is an adjustable and flexible method to edit structures and correct the mistakes caused by other programs (here are Tekla and RFEM). The integration method should still be developed. In the ideal case, the "full integration" type should be implemented in the projects. Nevertheless, the method makes possible that users themselves set the integration system according to their

specific demands. Once the system is developed, it or its part can be implemented with other structures. Moreover, the integration system can be extended to involve more programs for utilized in structural design as Mathcad, Excel, ArchiCAD etc. To conclude, the integration between structural software is worth to be developed and at least for Tekla Structures and RFEM connection, Rhinoceros with Grasshopper is the method providing satisfactory results.

REFERENCES

A-Insinöörit. (2018, December 13). A-Insinöörit valittiin Tampereen Kansi -hankkeen pohjoiskannen suunnittelijaksi – suunnittelussa hyödynnetään algoritmeja. Retrieved February 27, 2019, from A-INSINÖÖRIT: https://www.ains.fi/uutiset/insinoorit-valittiin-tampereen-kansi-hankkeen-pohjoiskannen-suunnittelijaksi-suunnittelussa-hyodynnetaan-algoritmeja/

Bucci, F., & Mulazzani, M. (2002). Luigi Moretti: Works and Writings. New York: Princeton Architectural Press .

Carrasco, O. (2019, 1 24). Grasshopper-Tekla live link applied to large scale architectural projects, or what do have in common Airports and Basilicas. (O. Carrasco, Performer) Aalto university, Helsinki, Southern Finland, Finland.

davidaewen. (2015, September 18). japan,kitagata. Retrieved February 27, 2019, from Flickr Hive Mind: https://hiveminer.com/Tags/japan%2Ckitagata

Dlubal Engineering Software. (2010). Program RX-Tekla. Tiefenbach: Dlubal Engineering Software.

Dlubal Software Engineering. (2009). Program RFEM 4. Tiefenbach: Dlubal Software Engineering.

Dlubal Software GmbH. (2016, December 8). Member Type. Retrieved February 26, 2019, from Structural Analysis and Design Software: https://www.dlubal.com/en/solutions/online-services/glossary/000016

Dlubal Software GmbH. (2017, March 17). Determining and Using Effective Lengths. Retrieved March 7, 2019, from Structural Analysis and Design Software: https://www.dlubal.com/en/support-and-learning/support/knowledge-base/001288

Dlubal Software GmbH. (2017, February 1). Manual RFEM 5. Retrieved October 22, 2018, from Structural Analysis and Design Software: https://www.dlubal.com/en/downloads-and-information/documents/manuals?filter=all

Dlubal Software GmbH. (2018, June 14). RFEM - FEM Structural Analysis Software. Retrieved February 28, 2019, from Structural Analysis and Design Software: https://www.dlubal.com/en/products/rfem-fea-software/what-is-rfem

Dragicevic, P. (2018, October 20). 1889 – Gaudí's Hanging Chain Models. Retrieved February 27, 2019, from List of Physical Visualizations and Rlated Artifacts: http://dataphys.org/list/gaudis-hanging-chain-models/

Finnish Standards Association SFS. (2005). Eurocode 3: Design of steel structures. Part 1-1: General rules and rules for buildings. Brussel: Finnish Standards Association SFS.

Fundació Junta Constructora del Temple Expiatori de la Sagrada Família. (2018, November 21). Double twist columns: geometry, mechanics and symbolism. Retrieved February 26, 2019, from Blog Sagrada Família: https://blog.sagradafamilia.org/en/specialists/double-twist-columns/

Hernandez, C. R. (2006, May). Thinking parametric design: introducing parametric Gaudi. Design Studies, 27(3), 309-324.

Hibbeler, R. C. (2010). Mechanics of Materials (8th ed.). Upper Saddle River, N.J: Pearson Prentice Hall.

Makert, R.;& Alves, G. (28. November 2016). Between Designer and Design: Parametric Design and Prototyping Considerations on Gaudí's Sagrada Familia. Periodica Polytechnica Architecture, ss. 89-93.

McNeel. (18. December 2018). Rhino User Meeting Helsinki. Haettu 18. December 2018 osoitteesta Rhino News, etc.: http://blog.rhino3d.com/2018/12/rhino-user-meeting-helsinki-24th.html

McNeel Europe. (2019, January 22). food4Rhino. Retrieved February 27, 2019, from food4Rhino: https://www.food4rhino.com/

Millais, M. (2017). Building structures : understanding the basics (3rd ed.). Abingdon: Routledge.

Mode Lab. (2018, August 2). Grasshopper Primer. Retrieved November 12, 2018, from MODELAB: https://www.modelab.is/grasshopper-primer/

Oxman, R. (2006). Theory and design in the first digital age. Design Studies, 229-265.

Pirhonen, I. (2019, 3 4). Integrated Design. (M. Ovanesova, Interviewer)

Prins, M., & Owen, R. (2010). Integrated Design and Delivery Solutions. Architectural Engineering and Design Management Ser., 81.

Robert McNeel & Associates. (2018, October 25). What are NURBS? Retrieved February 27, 2019, from Rhinoceros: https://www.rhino3d.com/nurbs

Robert McNeel & Associates. (18. Agust 2018). What's New in Rhino 6 for Windows. Haettu 27. February 2018 osoitteesta Rhinoceros: https://www.rhino3d.com/6/new

Sakamoto, T., & Ferré, A. (2008). From Control to Design. Barcelona: Actar.

Simões da Silva, L., Simões, R., & Gervásio, H. (2010). Design of Steel Structures: Eurocode 3 - Design of Steel Structures. Part 1-1 - General Rules and Rules for Buildings. Berlin: ECCS, Ernst & Sohn.

Trimble Inc. (2018, November 28). Company History. Retrieved February 28, 2019, from Trimble: https://www.trimble.com/Corporate/About_History.aspx

Trimble Solutions Corporation. (2016, April 1). Tekla Structures 2016, Analysis. Retrieved March 17, 2019, from Tekla Structures: https://teklastructures.support.tekla.com/system/files/manual/Analysis.pdf

Trimble Solutions Corporation. (2018, December 6). Tekla Structures - the Most Advanced BIM Software for Structural Workflow. Retrieved February 27, 2019, from Tekla: https://www.tekla.com/products/tekla-structures

Trimble Solutions Corporation. (2019, February 13). HOME. Retrieved February 28, 2019, from Tekla Campus: https://campus.tekla.com/

Wyatt, K., & Hough, R. (2013). Principles of Structure, Fifth Edition. Stroud: CRC Press.