AALBORG UNIVERSITY

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

APPENDIX

Created by
Tamás Drávai
Haroon Khalyar
Gábor Nagy

Supervised by
Kjeld Svidt
Lars Pedersen

June 7, 2016
Appendix A

Interview questions

1. How does the usual design process look like at your firm?

2. Do you use any CAD software (Revit, Tekla, ArchiCad, etc.) during that process?
   (a) If yes, then what kind and to what extent? What is the reason for choosing this software?
   (b) How did the transition into using CAD software in your company influence your work flow?
   (c) If no, why don’t you use CAD software?
      i. Do you plan to implement CAD software in the future?
      ii. What do you see as the biggest obstacle for you to implement CAD software into the design process?

3. What kind of training did you get for CAD software usage and integration?

4. How does the CAD software integration differ for different project sizes?

5. What issues are you experiencing when you use CAD-FEM data exchange in your projects?

6. Who owns the CAD model? (architects, engineers, constructor, etc.)

7. Have you encountered issues with copyright of model content?
   (a) If yes, then what kind of issues were they, and which are the ones that occur most often?
   (b) How do you usually solve these issues?

8. Do you think, that a more streamlined data exchange process between CAD and FEM software can start or increase the integration of CAD and FEM software interoperability into the design process?

9. What kind of information do you expect a CAD model to contain?

10. What kind of FEM software do you use?

11. What kind of data do you use from the CAD model for FE analysis?
(a) Can you get all the necessary data for the analysis from the CAD model?
   i. If no, then what type of data is missing most often?

12. What is the most time consuming part of the exchange process between CAD and
    FEM software?

13. Have you experienced problems with this interoperability?
   (a) If yes, describe the issues you encountered and your solution to these issues.

14. What kind of file formats do use for the exchange?

15. How do you communicate with the CAD modeller?

16. What kind of tools and information do you exchange with the CAD modeller?

17. What kind of software do you use for detailing?

18. How do you integrate the structural analysis results into the detailing process?

19. What do you see as the worst developed aspect of the structural design process?

20. Do you have any additional comments that is important to consider regarding the
    integration process between CAD and FEM?
Used software

B.1 CAD software

B.1.1 Autodesk Revit 2016

Version: 16.0.490.0 (x64)

Revit is a robust BIM software, with various fields of application such as architecture, structural engineering, MEP, construction management, etc. Regarding the topic of the project, Revit has a very powerful direct link with Autodesk Robot Structural Analysis (RSA), can export-import data with FEM-Design by the StruXML file format, and has great support and customizability for the IFC standards. It is also possible to export files with sat format, which is a file format supported by a number of FEM software.

B.1.2 Autodesk AutoCAD 2015

Version: J.141.0.0 (x64)

AutoCAD is also a robust CAD software. The software itself is very capable for drawing in 2D and 3D. With different expansion applications, the software can be expended with architecture, mechanical, infrastructure, electrical, GIS and structural detailing. The latter, as ProSteel, is used in this project. AutoCAD uses dwg file format, moreover, it is capable to export sat file format, which can be used, for example in Abaqus/CAE for FE analysis.

B.1.3 Tekla Structures 21.1 Service Release 1

Version: 17703 (x64)

Tekla Structures Learning is also a very capable BIM software and especially useful for steel structures. It has a well-developed data exchange link with FEM-Design through StruXML and RFEM through a direct link, while also supporting IFC export-import with a high level of customizability. Furthermore, it is possible to export and import stp files.

B.1.4 Bentley AECOsim Building Designer V8i

Version: 08.11.09.831 (x32)
AECOsim is a complex and powerful BIM software with six independent design modules. These modules are architectural, structural, mechanical, HVAC, plumbing and electrical module, can be run separately, but inside a specific project all modules are available for the user. AECOsim was mostly created for American design codes, but with different expansion packs the software can be used with international codes also, including the Eurocode. All Bentley software have a direct link with each other which is called Integrated Structural Model (ISM). This direct link enables to export-import data, and it also supports IFC. Moreover, AECOsim also supports stp files.

B.1.5 ProStructures for AutoCAD V8i

Version: 08.11.11.132 (x64)

ProStructures is an add-in to AutoCAD developed by Bentley. It contains two independent modules, which are ProSteel and ProConcrete and together the two modules form ProStructures. In this project only ProSteel is used. In ProSteel the user has opportunity to create basic and advanced 3D models, 2D design plans and advanced connections. Since, the software is a Bentley product the main data exchange link is the ISM.

B.2 FEM software

B.2.1 Autodesk Robot Structural Analysis 2016

Version: 29.0.4.5700 (x64)

Autodesk RSA is a widely used and versatile structural analysis software. Aside from FEM it can handle a variety of code based design and simulations. As mentioned earlier it has a very powerful direct link with Revit, but it also supports a number of file formats e.g. IFC files (import), stp files (import and export), sat files (import and export), etc. It does not support StruXML.

RSA is used for modeling, analysing and designing different types of structures. The user creates a model and can carry out structural analysis to verify obtained results or perform code check calculations. Furthermore, the user is able to prepare project documentation for a designed structure.

In RSA the user can do linear, non-linear according to Autodesk Inc. [2016e] and dynamic structure analysis according to Autodesk Inc. [2016a]. Plasticity is included in the design of reinforced concrete beams or columns c.f. Autodesk Inc. [2016c]. For steel structures the user can choose elastic-perfectly plastic material model or elastic-plastic material model with strain hardening c.f. Autodesk Inc. [2016b]. Furthermore, the user can also check for lateral torsional buckling. The software is capable of meshing a structure automatically,
and the user can refine the mesh in selected areas. Available meshing options include both shell and solid according to Autodesk Inc. [2016d], and supported type of finite elements include 4-node quadrilaterals, 4-node tetrahedrons, etc.

In RSA the user can work with an array of 70 built-in design codes, including Eurocodes according. [Autodesk Inc., 2016f].

B.2.2 FEM-Design 15

Version: 15.00.002 (x64)

FEM-Design is a frequently used FEM software that is also capable of code based design. It can exchange data both ways with Revit and can import from Tekla Structures Learning through the StruXML file format. At the time of this writing it only allows for the import of IFC files, however the export feature will possibly be added in the future. Furthermore, FEM-Design also supports import and export of dwg files.

The model is created in 3D within the software or imported from CAD-software. It is possible to add structural loads that include point, line, surface, temperature etc. Furthermore, the software can generate wind and snow loads automatically according to building location and code. The finite element mesh is generated and optimized automatically. It is possible to use both solid and shell elements e.g. 4-node quadrilaterals, 4-node tetrahedrons, etc. The software is capable of doing linear, nonlinear and dynamic analysis. The results of the analysis can be presented using a number of methods, e.g. graphs, contour lines, colour palettes, etc. [StruSoft AB, 2016b]

Design process in FEM-Design is carried out according to Eurocode with national annexes for steel, concrete and timber. It has Auto Design feature that helps the user to choose the most efficient cross-section or reinforcement arrangement. Automatic utilization checks for stresses, flexural buckling, lateral torsional buckling and web and flange buckling is included in steel design. Concrete design includes applying required reinforcement to areas from a list of available diameters and spacings and the calculation includes plasticity based on Eurocode 2. Automatic utilization check for stresses is also included for timber design in addition to flexural and lateral torsional buckling check. Moreover, it is possible to create complete and structured documentation of projects in FEM-Design. [StruSoft AB, 2016a]

B.2.3 Bentley STAAD.Pro

Version: 20.07.11.5 (x32)

STAAD.Pro is a FEM structural analysis and design tool from Bentley and can handle steel, concrete, timber and aluminium. The user can create the model in STAAD.Pro in 3D environment or import the model from a CAD software. The software supports a number
of industry standards for interoperability e.g. IFC, ISM, CIS/2 etc.

*STAAD.Pro* is capable of doing linear, nonlinear and dynamic analysis. Furthermore, it has a functionality called The Advanced Mesher that creates a mesh automatically and allows the user to check the quality of it visually. *STAAD.Pro* is equipped with both shell and plate finite element and supports 4-node elements, 8-node elements, etc. Furthermore, *STAAD.Pro* is also capable to check for instability e.g. lateral torsional buckling capacity of a structure. [Bentley Systems, Inc., 2016b]

Design of structures in *STAAD.Pro* is carried out according to design codes, and the software supports over 80 international codes including US, European (including Eurocodes), etc. Bentley Systems, Inc. [2016a].

### B.2.4 RFEM

Version: 5.06.1103 (x64)

*RFEM* is a structural FEA software from Dlubal Software GmbH. It is capable of structural and dynamic analyses in steel, reinforced concrete and timber structures. The software has a modular software structure, so the user can download individual Add-on Modules for specific purposes. *RFEM* can exchange data with other softwares using either direct link (*AutoCAD*, *Tekla Structures*) or by international standards e.g. IFC, ISM, SAT, etc.

Additionally, with *RFEM* the user can design member, plate, wall, folded plate, shell and solid elements. The size and shape of finite elements (triangles or quadrilaterals) can be selected by the user, who also has the option to set local mesh refinements.

*RFEM* is a multilingual software and supports national and international standards, including Eurocode with national annexes. It supports linear static, second order and large deformation analysis. Furthermore, there is an option to consider non-linear effects such as failure of tension members, plastic hinges, slippage, etc. Moreover, the user can perform stability analyses e.g. buckling (reduced stress, effective width, second order analysis).

### B.2.5 Abaqus/CAE

Version: 6.14-2 (x64)

*Abaqus/CAE* is a powerful and versatile tool for advanced FE analyses and simulations. It is capable of dealing with linear and non-linear problems, complex material models and assemblies among many other advanced features. It also allows for a scripting operation and a high-level of customization for calculation properties to optimize the analyses for the available computational resources.

Regarding the data exchange with other applications, *Abaqus/CAE* can handle direct data
B.2. FEM software

exchange with various CAD applications like CATIA V5, SolidWorks and Pro/Engineer. It also supports several different types of model geometry import formats like stp, SAT or IGES.

With regards to the analysis capabilities of Abaqus/CAE, there is a wide array of finite element types. In addition to beam elements, there are various types of shell, membrane and solid elements. The calculation properties, like number of integration points, for each element type can be fine tuned for a specific problem and computer.

A detailed description of the features and technical capabilities can be found in Systèmes [2014].
Data exchange description

C.1 Revit - RSA direct link

After the model was prepared, the data transfer could be initiated by a specified button in the user interface, which can be seen in Figure C.1.

![Data transfer user interface in Revit](image)

It can be seen that it allows for sending the model to RSA and also to update the current model in RSA. It is also possible to choose whether to send it directly (which requires that the two applications are installed on the same computer) or to an intermediate file, which is quite useful as in most cases the Revit model and the structural analysis are done by two different people. However, in this project both tasks were handled by the same person, thus the direct link was used. Another useful possibility is to choose to only send parts of the model for structural analysis, which can be used in many practical scenarios because, depending on what the user is analysing, many elements that are modeled in Revit are not relevant for the structural analysis.

Also there are cases where only certain parts of a structure are to be analysed. The transferred model in RSA can be seen in Figure C.2.
C.2 StruXML

C.2.1 Tekla Structures Learning - FEM-Design connection

After the model was prepared (defined section profile, material properties, boundary conditions) the next step was to create an analytical model of either the whole model or a selected part, where it was also possible to perform design calculations if an analysis application was installed and linked to Tekla Structures Learning e.g. SAP2000 (this was not the case in this project). The first step is illustrated in Figure C.3.

Before the model was exported, it needed to be prepared. Only analytical model categories could be exported from Revit to FEM-Design, so all elements that the user wants to import need to have an analytical model enabled. Furthermore, it was assured that the model was consistent (the elements were connected with each other) and each element had a valid structural material assigned to it.
C.2 StruXML

After creating the analysis model, the next step was to export the model to *FEM-Design* through *Tekla StruXML Export*. It was only possible to create the StruXML file while *Tekla Structures Learning* and *Tekla StruXML Export* were running simultaneously. The interface of *Tekla StruXML Export* is illustrated in Figure C.4.

In this window all the necessary mapping was done for materials and cross-sections. The user needed to find the corresponding material or cross-section in *FEM-Design* library and map it to the one from the *Tekla Structures Learning* library, e.g. the steel beams material was defined as S235JR in *Tekla Structures Learning*. This was not recognised in *FEM-Design*, so it was necessary to map this material to S 235 as it was defined in *FEM-Design*.
C.2. StruXML

The process of mapping is shown in Figure C.5. In a similar manner the cross-sections were mapped so it was recognized once the model was exported to *FEM-Design*.

![Figure C.5: Mapping of materials for the steel beam.](image1)

Once the mapping was done, the model was exported to a StruXML file, which could be opened in *FEM-Design*, like shown in Figure C.6.

![Figure C.6: The imported StruXML file in FEM-Design for the steel beam.](image2)

C.2.2 Revit - FEM-Design connection

Once *StruSoft StruXML Revit Add-in* was installed, it appeared in *Revit* under 'Analyse' tab, which is illustrated in Figure C.7.

![Figure C.7: The installed StruXML Revit Add-in.](image3)
As it can be seen in Figure C.7 there is an option to both import and export StruXML files, which makes it possible to import a model from Revit to FEM-Design and, in contrast to the link between Tekla Structures Learning and FEM-Design, also export a model from FEM-Design to Revit allowing to update the model in Revit and the model in FEM-Design.

In the transferring process it was necessary to choose which code standard the design was based on. The interface of StruSoft StruXML Revit Add-in is shown in Figure C.8. In this case 'Eurocode (NA: Danish)' was chosen. Next, materials and sections were mapped to appropriate materials and sections in the FEM-Design library, like shown in Figure C.8.
After the mapping was done, the analytical model was simply exported to a known location on the user’s PC. In order to open the StruXML file, the user simply had to open a new project in *FEM-Design* and then load the StruXML file. The imported model in *FEM-Design* is shown in Figure C.9.

![Figure C.9: The imported model in FEM-Design from Revit.](image)

C.3 ISM

C.3.1 AECOsim - STAAD.Pro

Figure C.10 shows the ISM interface in *AECOsim*. Also, a small description is provided for better understanding.
As it can be seen on the figure above, the user can choose from four options, depending on if the task is to export or import. In this case only the first two options are described because the third and the fourth option have less importance in this project.

The first option on the left is called Create ISM repository, which means exporting the current project which is created by the user. The second option is New from ISM repository. The meaning of this to import a project which is made in a different software. The repository means the entire project what the user can see on the display and also contains the relevant informations for the project, which are to be exported or imported.

When the data was to be exported from *AECOsim* to *STAAD.Pro* a simple acceptance of the elements was needed (in this case only one beam was transported to *STAAD.Pro*), which can be seen on figure C.11.
C.3. ISM

Figure C.11: Accepting the transformation in AECOsim.

After the ISM repository was created, it could be imported to the FEM software. At this point of the process, changes could be made on the data, which can also be seen on the figure below.

Figure C.12: Importing repository into FEM software.
During the import process the *Structural Synchronizer* software showed the user the model, which was to be imported. At this stage a decision could be made whether the whole model needed to be imported or only one or more parts of it. In this case, related to the project, only one beam was to imported to the FEM software. This stage of the process can be seen on figure C.13.

![Quick view of the model before validating the import.](image)

**Figure C.13:** Quick view of the model before validating the import.

After validating the import the model was ready to be used for structural analysis. The result of the process can be seen on figure C.14.

![Imported model in FEM software.](image)

**Figure C.14:** Imported model in FEM software.
C.3. ISM

C.3.2 ProSteel - STAAD.Pro

The transfer process and the four icons were described in Section 4.3 (p. 21). Once the model is ready to be exported, the user simply has to select the corresponding icon. This can be seen on Figure C.15. The icons can be found on the main user interface of ProSteel.

![ISM link between ProSteel and STAAD.Pro.](image)

When the user chooses the option to create the ISM file, the Structural Synchronizer opens and the user can decide whether only part of the model or the entire model is to be exported. Because, this process was already described and visualized in Section 5.1.4 (p. 38), no further discussion is provided for this part.

C.3.3 ProSteel - RFEM

The process is simple and straightforward. The user has to open a new project in RFEM and after this the import option can be chosen. At the import option the user can select several options and formats. In this case the ISM format has to be selected which can be seen on Figure C.16.

![Import variations in RFEM.](image)

Figure C.15: ISM link between ProSteel and STAAD.Pro.

Figure C.16: Import variations in RFEM.
When the file was selected and accepted the program offers another option for the user, which is, either to import the physical or the analytical model. In this case the analytical model is selected. This can be seen on the figure below.

![Import Options](image)

Figure C.17: Possibility of choice to import physical or analytical model in RFEM.

When the user decides about the physical or analytical model, the import process finishes and the model can be used for further analysis.

C.4 IFC

C.4.1 Revit

For Revit there was a separate extension, which expanded its export capabilities significantly. It allowed for a very high level of customization and had many predefined MDVs that were easy to use to modify certain elements of them to fit a specific scenario. This extension was downloaded from the Autodesk App Store for free. Figure C.18 shows the user-interface for the IFC export.
To create the IFC file properly, meaning that the required information is exported to it in an accurately defined data structure, the *Revit* objects needed to be mapped to the appropriate IFC entities. This was done in the IFC export settings which is shown in Figure C.19. *Revit* has a default setting for this, but it should be examined and if needed adjusted to the specific project.

Aside from the various included MVDs, it was also possible to create a user-defined MVD based on one of the predefined ones, which also allowed for the inclusion of custom property sets. This could be done with a specific txt file that had to be created manually by the user. This txt file specified the information that the user wanted to export and mapped the *Revit* property to a suitable IFC entity.
C.4.2 Tekla Structures Learning

In *Tekla Structures Learning*, the method to create the IFC file was quite similar to *Revit*, however it did not have the same amount of predefined MVDs. The export could be done in the IFC export interface window, shown in Figure C.20.

![IFC export interface in Tekla Structures Learning.](image)

Figure C.20: IFC export interface in *Tekla Structures Learning*.

As with *Revit*, it was also possible to create custom property sets with *Tekla Structures Learning*, allowing for the inclusion of user defined attributes for each object. Every supported property can be chosen from a separate window, where the attributes can be individually modified and assigned to an IFC entity. It can be seen in Figure C.21.

![IFC mapping interface in Tekla Structures Learning.](image)

Figure C.21: IFC mapping interface in *Tekla Structures Learning*. 
C.4. IFC

C.4.3 Bentley AECOsim Building Designer

As in Revit and Tekla Structures Learning, AECOsim also has the option to export and import IFC. The software only supports IFC2x3 format, as it can be seen on figure C.22.

![IFC Export dialog box](image)

Figure C.22: IFC exporting in AECOsim.

As with the previous software, mapping of the different objects was also done in AECOsim. The difference in AECOsim was, that the object mapping was done automatically by the program itself. Although, there was an opportunity to do it manually, this option was really complicated, therefore a good overall insight of the software is recommended. Also, at this time manual mapping was limited, only five attributes (reference, slope, span, load-bearing and fire rating) could be modified. This is shown on figure C.23.
C.5 Tekla Structures Learning - RFEM direct link

It is important to note here, that at STAAD.Pro the import of the IFC was different than for the other software investigated in this project. STAAD.Pro was the only exception where the IFC file had to be converted into an ISM file. After this, the ISM file could be imported into STAAD.Pro for further analysis.

C.5 Tekla Structures Learning - RFEM direct link

There are several data exchange options between Tekla Structures Learning and RFEM. In the following the import method is described, but it is also possible to export the model from RFEM to Tekla Structures Learning and thus updating the model. In this project the direct import from Tekla Structures Learning is used to import the analytical model of the structure. During the transfer the user can either click on the 'Direct import from Tekla Structures’ or go into 'File’ and 'Import’, where the window shown in Figure C.24 will appear.
As it can be seen, RFEM supports several data exchange formats. Under 'Direct Imports' Tekla Structures is chosen and the user is presented with the following window with import options c.f. Figure C.25. In this window the user can choose either 'Physical Model' or 'Analytical Model'. It is also possible to tick of several options e.g. 'Import load cases' and 'Set Z-axis upward' (if this is not ticked the z-axis will be set downwards).

The user can also define conversation files for cross-section and material. This is done under 'Detail settings' shown in Figure C.26 during the import process.
C.6 CIS

In *Tekla Structures Learning* the user can choose to export two types of model: the 'Analysis Model' or the 'Design/manufacturing Model' (Figure C.27).

![Figure C.27: Export from *Tekla Structures Learning* using CIS.](image)

In this case 'Analysis Model' is chosen. The user is then presented with the following window, shown in Figure C.28.

![Figure C.26: Detailed import settings.](image)
The user needs to choose CIS version, in this case CIS/2 was chosen, since this is what 
STAAD.Pro is compatible with. In step file box, the user needs to write the stp file title, 
which is the file imported into STAAD.Pro. From the flavour list, European standard was 
chosen. Furthermore, 'Yes' option was chosen in 'Split members' box. For example a beam 
may be connected to three columns, so that one of the columns supports the beam in the 
middle and the other two in each end of the beam. By enabling 'Split members' the beam 
is split into two equal elements in the CIS model. If 'No' is selected there will be just one 
beam element with two nodes; one in each end.

C.7 Tekla Structures Learning - Abaqus/CAE

The transfer procedure is initiated by 'File' and 'Export' in Tekla Structures Learning. 
Then, the user needs to select 'DWG/DXF...' as shown in Figure C.29, and is presented 
with the window in Figure C.30.
In the export settings menu, as shown in Figure C.30, 'Export as Faces' were chosen with high part accuracy.

Other options available are 'Lines' (exports parts as lines located in the center of the profile cross-section), 'Center lines' (exports parts as center lines) and 'Reference lines' (exports parts as reference lines, drawn between the creation points). In this case, 'Export as Faces' were best suited.

The frame corner created in Tekla Structures Learning is imported as a polyface mesh into Abaqus/CAE. The user needs to geometrically mesh the imported part, as shown in Figure C.31. After meshing the frame corner, the user needs to convert the geometrical mesh to a 3D solid by 'Convert to Solid' feature.
To geometrically mesh the part, mesh options shown in Figure C.32 were used. The settings were not changed from the default values aside from mesh type and mesh distance from original face.

The meshing part is only necessary in order to change the polymesh face to 3D solid in AutoCAD. Finally, the different parts of the frame corner were combined to just one part by use of 'Solid, union' under 'Solid' tab.

Once the frame corner is exported to Abaqus/CAE, it is easier to deal with just one part instead of several parts. In the latter case the user needs to assemble the different parts to
one part in *Abaqus/CAE*, define constraint between the different parts and assign material and cross-section for each part separately.
In the process of converting solid to shell, firstly 'Assign Midsurface Region' feature is used, where the whole model is selected as a region. Then, 'Offset Faces' feature was used to off-set the faces of the top flange to just one shell part at the center of the solid part. And same procedure with the bottom flange and the web. After off-setting faces of upper- and bottom flanges and the web, it was necessary to use 'Extend Faces' feature, since there was gaps between the flanges and the web. The web was extended to the flange in each side. Moreover, it was necessary to use use 'Remove Faces', since the edge of the web in the upper part of the frame was not aligned with the edge of the web in lower part, c.f. Figure D.1.

Figure D.1: Use of 'Remove Faces' feature in Abaqus/CAE.
List of external appendixes

The external appendix consists of a DVD with the following content.

1. Simple Cases
   - Beam FEM-Design
   - Beam Revit
   - Beam RSA
   - Beam Tekla Structures Learning
   - Frame FEM-Design
   - Frame Revit
   - Frame RSA
   - Frame Tekla Structures Learning

2. Advanced Cases
   - Conveyor bridge from 3D Structural Design
   - Platform structure from ISC
   - Fatigue analysis
   - Buckling analysis